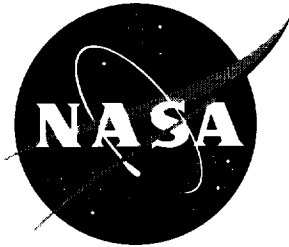


NASA/TM-2001-210838



Improved Equivalent Linearization Implementations Using Nonlinear Stiffness Evaluation

*Stephen A. Rizzi and Alexander A. Muravyov
Langley Research Center, Hampton, Virginia*

March 2001

The NASA STI Program Office ... in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counter-part of peer reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

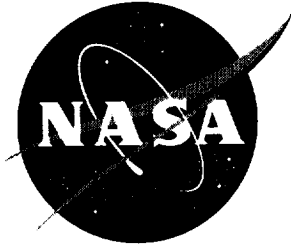
- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that help round out the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at <http://www.sti.nasa.gov>
- E-mail your question via the Internet to help@sti.nasa.gov
- Fax your question to the NASA Access Help Desk at (301) 621-0134
- Phone the NASA Access Help Desk at (301) 621-0390
- Write to:
NASA Access Help Desk
NASA Center for AeroSpace Information
7121 Standard Drive
Hanover, MD 21076-1320

NASA/TM-2001-210838



Improved Equivalent Linearization Implementations Using Nonlinear Stiffness Evaluation

*Stephen A. Rizzi and Alexander A. Muravyov
Langley Research Center, Hampton, Virginia*

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

March 2001

The use of trademarks or names of manufacturers in the report is for accurate reporting and does not constitute an official endorsement, either expressed or implied, of such products or manufacturers by the National Aeronautics and Space Administration.

Available from the following:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 487-4650

Foreword

This report documents two new implementations of equivalent linearization for solving geometrically nonlinear random vibration problems of complicated structures. The implementations are given the acronym ELSTEP, for “Equivalent Linearization using a STiffness Evaluation Procedure.” Both implementations of ELSTEP are fundamentally the same in that they use a novel nonlinear stiffness evaluation procedure to numerically compute otherwise inaccessible nonlinear stiffness terms from commercial finite element programs. The commercial finite element program MSC/NASTRAN (NASTRAN) was chosen as the core of ELSTEP. The FORTRAN implementation calculates the nonlinear stiffness terms and performs the equivalent linearization analysis outside of NASTRAN. The Direct Matrix Abstraction Program (DMAP) implementation performs these operations within NASTRAN. Both provide nearly identical results. Within each implementation, two error minimization approaches for the equivalent linearization procedure are available – force and strain energy error minimization. Sample results for a simply supported rectangular plate are included to illustrate the analysis procedure.

The majority of this work was originally performed by Alexander A. Muravyov during his tenure as a National Research Council post-doctoral research associate at the NASA Langley Research Center. Additional enhancements and interfaces were subsequently developed by Stephen A. Rizzi. The authors wish to thank Jay H. Robinson and Travis L. Turner of the Structural Acoustics Branch at the NASA Langley Research Center for helpful discussions and comments.

Table of Contents

1. Introduction	1-1
2. Finite Element Model Development	2-1
2.1. Normal Modes Analysis Bulk Data File	2-1
2.2. Static Analysis Bulk Data File	2-5
2.3. Random Analysis Bulk Data File	2-6
3. FORTRAN Implementation	3-1
3.1. Input Files	3-1
3.1.1. Modes Selection File	3-1
3.1.2. Spectral Density Loads File	3-1
3.1.3. Damping Matrix File	3-3
3.1.4. Parameter File	3-3
3.2. Calculation of Nonlinear Stiffness Coefficients	3-4
3.3. Equivalent Linearization Procedure	3-7
3.3.1. Convergence	3-10
4. DMAP Implementation	4-1
4.1. Input Files	4-1
4.1.1. Modes Selection File	4-1
4.1.2. Spectral Density Loads File	4-2
4.1.3. Damping Matrix File	4-3
4.1.4. Parameter File	4-3
4.2. Calculation of Nonlinear Stiffness Coefficients	4-4
4.3. Equivalent Linearization Procedure	4-7
4.3.1. Convergence	4-10
5. Post-Processing for Displacement, Stress and Strain PSDs	5-1
6. Example Problem	6-1
6.1. Nonlinear Stiffness Results	6-1
6.2. Equivalent Linearization Results – Force Error Minimization	6-2
6.3. Equivalent Linearization Results – Strain Energy Error Minimization	6-3
6.4. Post-Processing Results	6-3
7. Summary	7-1
8. References	8-1
Appendix A Software Distribution and Installation	A-1
Appendix B Listing of Sample Normal Modes Analysis Bulk Data File	B-1

Appendix C	Listing of Sample Nonlinear Static Analysis Bulk Data File.....	C-1
Appendix D	Listing of Sample Random Analysis Bulk Data File.....	D-1
Appendix E	Listing of Sample Bulk Data File “fixed103_eg.bdf”	E-1
Appendix F	Listing of Sample Bulk Data File “fixed106_st.bdf”	F-1
Appendix G	Listing of Sample Bulk Data File “fixed101_st.bdf”	G-1
Appendix H	Listing of Sample Bulk Data File “fixeD103_eg.bdf”	H-1
Appendix I	Listing of Sample Bulk Data File “fixeD106_st.bdf”	I-1
Appendix J	Listing of Sample Bulk Data File “fixeD101_st.bdf”	J-1

List of Figures

Figure 1: Screen capture of PATRAN "Analysis" dialog box for solution 103.....	2-1
Figure 2: Screen capture of PATRAN "Solution Type" dialog box for solution 103.	2-2
Figure 3: Screen capture of PATRAN "Solution Parameters" dialog box for solution 103.....	2-2
Figure 4: Screen capture of PATRAN "Subcase Create" dialog box for solution 103.	2-3
Figure 5: Screen capture of PATRAN "Subcase Parameters" dialog box for solution 103.	2-4
Figure 6: Screen capture of PATRAN "Output Requests" dialog box for solution 103.	2-4
Figure 7: Screen capture of PATRAN "Solution Parameters" dialog box for solution 106.....	2-5
Figure 8: Screen capture of PATRAN "Subcase Parameters" dialog box for solution 106.	2-6
Figure 9: Screen capture of PATRAN "Output Requests" dialog box for solution 106.	2-7
Figure 10: Screen capture of PATRAN "Solution Parameters" dialog box for solution 111.....	2-7
Figure 11: Screen capture of PATRAN "Eigenvalue Extraction" dialog box for solution 111.	2-8
Figure 12: Screen capture of PATRAN "Subcase Parameters" dialog box for solution 111.	2-8
Figure 13: Flow-chart for calculation of nonlinear stiffness coefficients in the FORTRAN implementation.....	3-6
Figure 14: Flow chart for calculation of total equivalent stiffness matrix and RMS displacements in the FORTRAN implementation.....	3-7
Figure 15: Flow-chart for calculation of nonlinear stiffness coefficients in the DMAP implementation.....	4-6
Figure 16: Flow chart for calculation of total equivalent stiffness matrix and RMS displacements in the DMAP implementation.	4-7
Figure 17: Flow chart for post-processing.	5-1
Figure 18: Comparison of displacement PSDs using linear and equivalent linear analyses.....	6-5
Figure 19: Comparison of stress PSDs using linear and equivalent linear analyses.	6-5

List of Tables

Table 1: Comparison of RMS displacements from equivalent linearization and post-processing.....	6-4
---	-----

1. Introduction

Several methods exist for the prediction of geometrically nonlinear dynamic structure response including perturbation, Fokker-Plank-Kolmogorov (F-P-K), Monte Carlo simulation and equivalent linearization techniques. Perturbation techniques are limited to weak geometric nonlinearities. The F-P-K approach [1, 2] yields exact solutions, but can only be applied to simple mechanical systems. Monte Carlo simulation is the most general method, but computational expense limits its applicability to rather simple structures. Finally, equivalent linearization methods [2-6] have seen the broadest application for prediction of geometrically nonlinear dynamic response because of their ability to accurately capture the response statistics over a wide range of response levels while maintaining a relatively light computational burden.

Implementations of equivalent linearization using finite element analysis have been limited to special purpose computer codes. This is largely due to the inaccessibility of the nonlinear stiffness quantities in commercial finite element applications. That changed when an equivalent linearization analysis was introduced into MSC/NASTRAN (NASTRAN) [7] as a Direct Matrix Abstraction Program (DMAP) Alter [8, 9]. In that implementation, the equivalent stiffness was obtained as the sum of the linear stiffness and three times the differential stiffness. This formulation was found to over-predict the degree of nonlinearity and produce non-conservative results. Over-prediction of nonlinearity can produce the undesirable result of structural designs incapable of withstanding the applied loads in an acceptable fashion.

An activity was recently undertaken to more accurately determine the equivalent stiffness through a novel approach [10, 11]. In it, the nonlinear stiffness coefficients from commercial finite element programs may be numerically extracted by solving a series of inverse linear and nonlinear static problems. While this approach is applicable to any commercial finite element program having a nonlinear analysis capability, NASTRAN was selected due to its widespread use in the aerospace industry. The use of this new approach in an equivalent linearization analysis has been validated against F-P-K [11] and numerical simulation analyses [12] for clamped-clamped beams.

This report documents two new implementations of the above approach. The implementations are given the acronym ELSTEP, for "Equivalent Linearization using a STiffness Evaluation Procedure." Both implementations are fundamentally the same in that they each use the stiffness evaluation procedure indicated above. The FORTRAN implementation calculates the nonlinear stiffness terms and performs the equivalent linearization analysis outside of NASTRAN. The DMAP implementation performs these operations within NASTRAN. Both perform NASTRAN normal modes (solution 103), linear static (solution 101) and nonlinear static (solution 106) analyses for the calculation of the nonlinear stiffness terms and provide nearly identical results. Optional post-processing is performed by substituting the total equivalent stiffness matrix in place of the linear stiffness matrix in a random (solution 111) analysis.

Within each implementation, two error minimization approaches for the equivalent linearization procedure are available – force and strain energy error minimization. Either or both may be run to obtain the total equivalent stiffness and root-mean-square displacements. The traditional force error minimization approach [3, 4] is implemented as described in [10, 12]. An extension of a single degree-of-freedom strain energy error minimization approach [5, 6] to multiple degree-of-freedom systems [10] is also implemented.

It should be noted that this analysis has been only developed for, and only validated against cases in which the structure exhibits stretching of the middle surface, e.g. clamped and simply supported structures. The reader is advised not to apply it to problems in which the nonlinear behavior is manifested in other ways, such as in cantilevers.

Information about the ELSTEP source code distribution and installation may be found in Appendix A. The FORTRAN implementation is platform independent. All FORTRAN programs, in both the

FORTRAN and DMAP implementations, should be compiled with a FORTRAN 90 compiler. The DMAP implementation uses string-based Alters of NASTRAN version 70.0.0 solutions. It is therefore expected to be upward compatible with future versions of NASTRAN.

Throughout this document, specific filenames used in the analysis are made reference to in **bold** font. Sample results for a simply supported rectangular plate are included to illustrate the analysis procedure.

2. Finite Element Model Development

In order to perform the analyses described in Sections 3 – 5, it is necessary to develop three bulk data files based on two finite element model databases. One database is for a dynamic model and one is for a static model. The dynamic model is used for normal modes (solution 103) and random (solution 111) analyses. The static model is a modification of the dynamic model and is used for linear static (solution 101) and nonlinear static (solution 106) analyses. In the following, it is assumed that the finite element models are built in MSC/PATRAN (PATRAN) [13], although this is not required.

2.1. Normal Modes Analysis Bulk Data File

- Create a new database called “*filename_eg*,” for example. Note that in the following, *filename* is an 8-character prefix.
- Create the finite element model as usual including boundary conditions, material properties, loading, etc.
- In the “Analysis” dialog box, type “*filename_eg*” as the job name. See for example Figure 1.

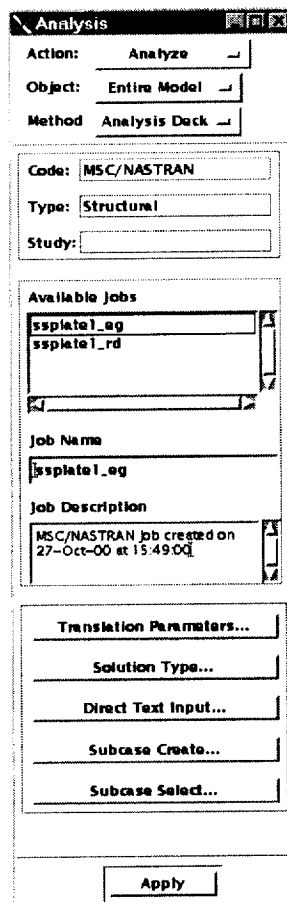


Figure 1: Screen capture of PATRAN "Analysis" dialog box for solution 103.

- In the “Solution Type” dialog box, choose “Normal Modes” analysis (solution 103), as shown in Figure 2.

- Set up the "Solution Parameters" dialog box as shown in Figure 3. The mass calculation may be lumped or coupled.

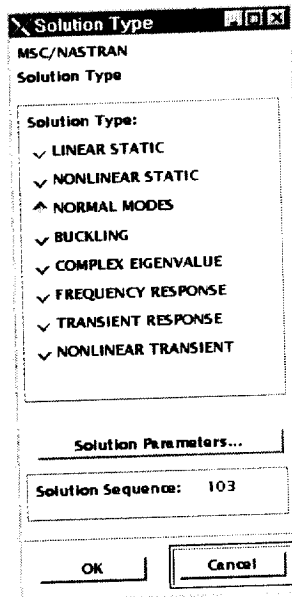


Figure 2: Screen capture of PATRAN "Solution Type" dialog box for solution 103.

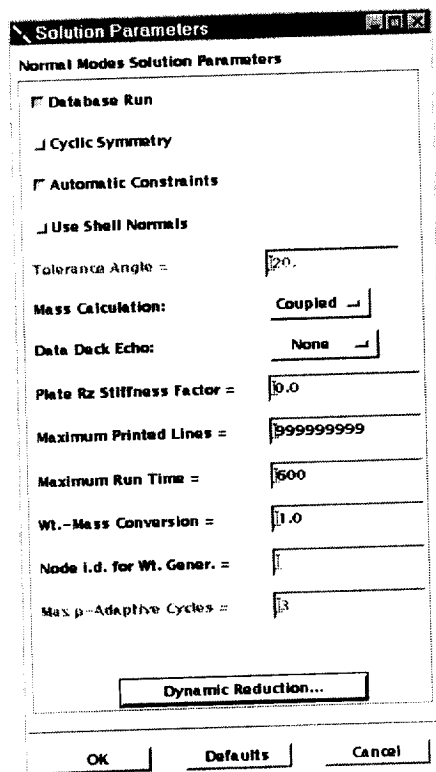


Figure 3: Screen capture of PATRAN "Solution Parameters" dialog box for solution 103.

- From the “Analysis” dialog box, select the “Subcase Create” dialog box and select the appropriate load subcase (one with boundary conditions but no loads), as shown in Figure 4.
- From the “Subcase Parameters” dialog box of the “Subcase Create” dialog box, set the options as shown in Figure 5, with the number of desired roots applicable for the particular analysis. This set of eigenvectors is subsequently referred to as the “large” set of eigenvectors.

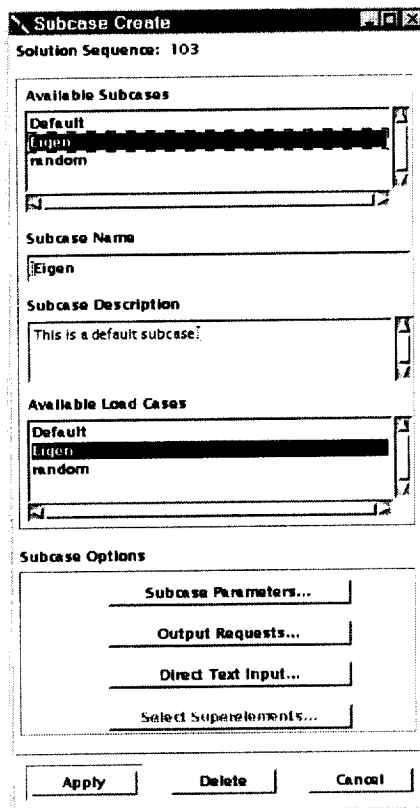


Figure 4: Screen capture of PATRAN "Subcase Create" dialog box for solution 103.

- From the “Output Requests” dialog box of the “Subcase Create” dialog box, choose the “Eigenvectors” result type as shown in Figure 6.
- Select the “Analysis Deck” method from the “Analysis” dialog box to create the file “filename_eg.bdf” for subsequent analysis. The extension “_eg” is required as it is referenced in the subsequent analyses. An example bulk data file is provided in Appendix B.

Subcase Parameters
REAL EIGENVALUE EXTRACTION

Extraction Method: Lanczos

Frequency Range of Interest

Lower =

Upper =

Estimated Number of Roots =

Number of Desired Roots =

Diagnostic Output Level:

Results Normalization

Normalization Method: Mass

Normalization Point =

Normalization Component:

Number of Modes in Error Analysis =

Figure 5: Screen capture of PATRAN "Subcase Parameters" dialog box for solution 103.

Output Requests

SUBCASE NAME: Default

SOLUTION SEQUENCE: 103

Form Type: Basic

Select Result Type

- Constraint Forces
- Multi-Point Constraint Forces
- Element Forces
- Element Strain Energies
- Element Strains
- Grid Point Stresses
- Grid Point Force Balance
- Eigenvectors

Output Requests

VECTOR(SORT1,REAL)=ALL FEM

Figure 6: Screen capture of PATRAN "Output Requests" dialog box for solution 103.

2.2. Static Analysis Bulk Data File

- Copy the dynamic database "*filename_eg.db*" to "*filename_st.db*," for example. The 8-character prefix *filename* must be the same as that used for the normal modes bulk data file.
- Open "*filename_st.db*" in PATRAN and make the following modifications:
 - Add an extra grid point with a connecting beam or rod element. Note the added element must have a nonlinear capability and be connected to the structure at a fixed boundary. Equivalence the node where the new element is connected to the structure and, if necessary, renumber so that the node numbers of the main structure are the same as those for the normal modes analysis. In this manner, the extra grid point and element number will be greater than the highest grid point and element number of the original model.
 - Create a new load case and delete any previously applied loads from the model. To the new load case, apply an arbitrary non-zero nodal force at the extra grid point.
 - To the new load case, create a constraint displacement set, which sets all degrees of freedom to zero values except for those at the extra grid point.

Note that the necessity to introduce an extra element is an artifact of the solver implemented in NASTRAN solutions 101 (linear static) and 106 (nonlinear static). Specifically, these solutions can only solve the forward static problem, that is, these solutions solve for a set of displacements from a specified loading. In order to determine the nonlinear stiffness coefficients, it is necessary to solve linear and nonlinear inverse problems, which compute the forces due to a prescribed set of displacements. The extra element is introduced to allow NASTRAN to solve for a dummy set of displacements with the set of prescribed displacements acting as displacement constraints. The sought vector of applied loads (not including the extra load) is identical to the set of single-point-constraint forces in NASTRAN terminology.

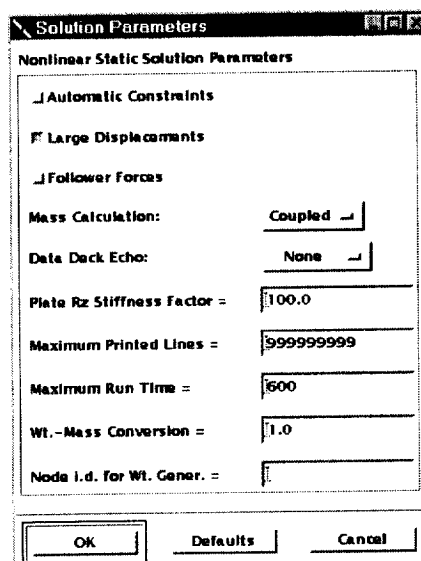


Figure 7: Screen capture of PATRAN "Solution Parameters" dialog box for solution 106.

- In the "Analysis" dialog box, type "*filename_st*" as the job name.

- In the “Solution Type” dialog box, choose “Nonlinear Static” analysis (solution 106). Note that the resulting “.bdf” file is also used in solution 101 and is automatically modified for this purpose, as described in Sections 3 and 4.
- Set up the “Solution Parameters” dialog box as shown in Figure 7. The mass calculation is irrelevant for the static case.
- From the “Analysis” dialog box, select the “Subcase Create” dialog box and select the new load subcase.
- From the “Subcase Parameters” dialog box of the “Subcase Create” dialog box, set the options as shown in Figure 8.

Subcase Parameters

Static Nonlinear Iterations

Number of Load Increments = 1

Matrix Update Method: Automatic

Number of Iterations per Update = 1

Allowable Iterations per Increment = 50

Convergence Criteria

☒ Displacement Error

Displacement Tolerance = 1.0e-03

☐ Load Error

Load Tolerance = 1.0e+13

☐ Work Error

Work Tolerance = 1.0e+13

OK Cancel

Figure 8: Screen capture of PATRAN "Subcase Parameters" dialog box for solution 106.

- From the “Output Requests” dialog box of the “Subcase Create” dialog box, choose the “Constraint Forces” result type as shown in Figure 9.
- Select the “Analysis Deck” method from the “Analysis” dialog box to create the file “filename_st.bdf” for subsequent analysis. The extension “.st” is required as it is referenced in the subsequent analyses. An example bulk data file is provided in Appendix C.

2.3. Random Analysis Bulk Data File

A random analysis (solution 111) is required only if post-processing of displacement power spectral densities or stress/strain root-mean-square values or PSDs is desired.

- In the “filename_eg.db” database, create a new load subcase and add the appropriate dynamic loading (in physical coordinates) corresponding to the modal loading specified in the equivalent linearization analysis (see Sections 3.1.2 and 4.1.2).

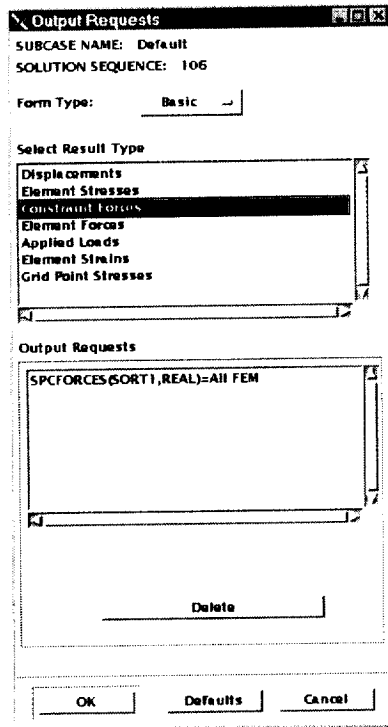


Figure 9: Screen capture of PATRAN "Output Requests" dialog box for solution 106.

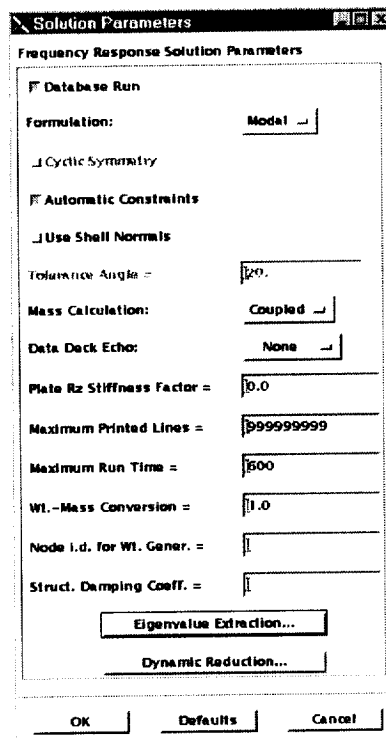


Figure 10: Screen capture of PATRAN "Solution Parameters" dialog box for solution 111.

- In the “Solution Type” dialog box, choose “Frequency Response” analysis.
- In the “Solution Parameters” dialog box, select the settings as indicated in Figure 10 with the formulation as modal to specify solution 111 and mass calculation as desired (lumped or coupled).
- Select the “Eigenvalue Extraction” dialog box from the “Solution Parameters” dialog box and set up the values as indicated in Figure 11. The number of desired roots should be the same as the large set of eigenvectors specified in the development of the normal modes analysis bulk data file.

Figure 11: Screen capture of PATRAN "Eigenvalue Extraction" dialog box for solution 111.

- From the “Subcase Create” dialog box, specify the frequency range in the “Subcase Parameters” dialog box. See Figure 12, for example. For consistent results between the equivalent linearization analysis and the solution 111 post-processing analysis, the frequency range should be the same as that used in the equivalent linearization analysis (see Sections 3.1.4 and 4.1.4).

Figure 12: Screen capture of PATRAN "Subcase Parameters" dialog box for solution 111.

- In the “Analysis” dialog box, type **fixeD111_rd** as the job name to create the bulk data file **fixeD111_rd.bdf**. This filename is required as it is referenced in the subsequent analysis.
- It is necessary to manually edit the bulk data file **fixeD111_rd.bdf** in preparation for the analysis run. The manual edits are indicated in bold in the sample listing provided in Appendix D.

3. FORTRAN Implementation

The analysis is subdivided into two parts. The first part calculates the nonlinear stiffness coefficients. The second part performs one of two equivalent linearization procedures to compute the total equivalent stiffness matrix and RMS displacements. A description of optional post-processing is provided in Section 5.

3.1. Input Files

In addition to the bulk data files discussed in Section 2, several additional input files are required to specify various parameters used in the analysis. These are detailed below.

3.1.1. Modes Selection File

The file **fixed.mod** indicates which eigenvectors out of the large set of eigenvectors participate in the analysis. The format for **fixed.mod** is a free ASCII format as specified below.

LINE 1: Number of selected modes (NMOD).

LINE 2: Selected modes (e.g. 1st and 4th).

LINE 3: Scaling coefficients for each eigenvector selected. The product of each eigenvector and scaling coefficient produces a displacement field, which is used as a prescribed displacement set. The scaling coefficients should be chosen to be equal and not made too large so that the product of the eigenvector and coefficient will represent a realistic prescribed displacement within the linear range. The value of 1.0e-4 appears to work well for cases considered thus far.

A sample listing of **fixed.mod** with two selected modes (NMOD=2) is provided below.

```
2
1      4
1.e-04  1.e-04
```

Note that specification of the participating modes may necessitate running a standard normal modes analysis prior to the procedure for calculating the nonlinear stiffness coefficients described in Section 3.2. In addition, if a post-processing analysis is to be performed (as described in Section 5), the same information must also be specified in the file **fixedD.mod**, as described in Section 4.1.1.

3.1.2. Spectral Density Loads File

The file **fixed.den** contains the spectral density matrix of the applied loads in modal coordinates $\tilde{S}_f(\omega)$. The format for **fixed.den** is a free ASCII format as specified below.

LINE 1: Number of breakpoints used to define frequency spectrum. (Min 2, Max 50). Between each breakpoint, each value of the $\tilde{S}_f(\omega)$ is linearly interpolated at the frequency increment specified in **fixed.par** (see Section 3.1.4).

LINE 2: Frequency of first breakpoint (Hz)

LINE 3: First row of modal spectral density matrix (NMOD entries) at first frequency breakpoint.

...

LINE 3 + NMOD:

Last row of modal spectral density matrix (NMOD entries) at first frequency breakpoint

(Lines 2 to Line 3+NMOD are repeated for each breakpoint)

A sample listing of **fixed.den** for a flat spectrum between 0-1024 Hz with two selected modes is provided below.

```

      2
0.000000000000000
6.83899171960566      2.27952175755947
2.27952175755446      0.759793205814123
1024.000000000000
6.83899171960566      2.27952175755947
2.27952175755446      0.759793205814123

```

To be explicit, at both 0 and 1024 Hz, the following spectral density matrix of applied loads in modal coordinates is specified:

$$\tilde{S}_{ff}(0,1024) = \begin{bmatrix} \tilde{S}_{ff11} & \tilde{S}_{ff12} \\ \tilde{S}_{ff21} & \tilde{S}_{ff22} \end{bmatrix} = \begin{bmatrix} 6.84 & 2.28 \\ 2.28 & 0.76 \end{bmatrix}$$

This matrix is computed as

$$\tilde{S}_{ff}(\omega) = \phi^T S_{ff}(\omega) \phi \quad (1)$$

where $S_{ff}(\omega)$ is the fully-populated, real-valued spectral density matrix of the load in physical coordinates and ϕ is the subset of selected eigenvectors. Because $S_{ff}(\omega)$ is real-valued, only correlated inputs are permitted. This condition allows loadings such as normal acoustic pressure, inertial loading (for base excitation), and one or more correlated point forces or moments. It does not allow loadings which generate complex terms, such as grazing incidence acoustic loading.

The values of $S_{ff}(\omega)$ specify the double-sided spectrum level in units²/rad/sec. In order to specify the same load in physical coordinates for a solution 111 post-processing analysis, the values of $S_{ff}(\omega)$ must be converted to a single-sided spectral density in units²/Hz by multiplying by $2 \times 2\pi$. These may then be specified on the TABRND1 card of the bulk data file.

From equation (1), it is obvious that a normal modes analysis must first be performed to obtain the eigenvectors. Several files containing eigenvectors are formed in the first part of the analysis used to calculate the nonlinear stiffness coefficients (see Section 3.2). The eigenvectors to be used in equation (1) are the set of selected g-size (all degrees of freedom including constrained ones) eigenvectors from the file **egveCa.dat**.

In order to facilitate generation of the modal spectral density matrix of the applied loading, the utility program **gensff** (described in Section 3.3) may be used to automatically compute the values and write them to **fixed.den** for a limited set of loading conditions.

3.1.3. Damping Matrix File

The file **fixed.dam** contains the diagonal modal damping matrix. The format for **fixed.dam** is a free ASCII formatted square matrix ($NMOD \times NMOD$) as specified below.

$$\begin{bmatrix} C_1 & 0 & \cdots & 0 \\ 0 & C_2 & 0 & 0 \\ \vdots & & \ddots & \\ 0 & & & C_{NMOD} \end{bmatrix}$$

A sample listing of **fixed.dam** with two selected modes is provided below.

```
14.6618    0.
0.         14.6618
```

In order to specify the same damping for a solution 111 post-processing analysis, the values must be converted to percent critical damping through the usual relation

$$\zeta_i = \frac{C_i}{2\omega_i}$$

where C_i are the modal damping values (as specified in **fixed.dam**) and ω_i are the linear eigenvalues, not those of the equivalent linear system. This value of load may then be specified on the TABDMP1 card of the bulk data file.

3.1.4. Parameter File

The file **fixed.par** contains various additional parameters required for the analysis. The format for **fixed.par** is a free ASCII format as specified below.

LINE 1: Minimum and maximum frequency range of the analysis (Hz). Note that the maximum frequency range should be selected well beyond the frequency of the highest selected mode. This is necessary so that the resonant frequencies don't shift out of the analysis bandwidth as nonlinearity increases. If the minimum or maximum analysis frequencies specified above fall outside of the bandwidth of applied loads (specified in **fixed.den**), the load will be padded with zeros at either low or high frequency end. If the minimum or maximum analysis frequencies fall inside of the bandwidth of applied loads, the loading will start and end at the frequencies specified above.

LINE 2: Frequency increment used in the analysis (Hz)

LINE 3: Weightings α and β . A discussion on how the weightings are used is given in [12]. The weightings must sum to 1. Typical initial values are $\alpha = 0.4$ and $\beta = 0.6$ for the force error minimization approach and $\alpha = 0.05$ and $\beta = 0.95$ for the strain energy error minimization approach. The choice of weightings may influence the solution to which the system converges, as discussed in Section 3.3.1.

LINE 4: Convergence criteria ϵ

A sample listing of **fixed.par** is provided below.

0. 1024.
0.25
0.4 0.6
0.005

3.2. Calculation of Nonlinear Stiffness Coefficients

The method for calculating nonlinear stiffness coefficients is shown in the flowchart in Figure 13. Each operation is separated by a dashed line and is run in the sequence shown from top to bottom. The various operations are described below.

- **prep103** – This FORTRAN program reads the bulk data file “*filename_eg.bdf*” and rewrites it to the file **fixed103_eg.bdf**. The program **prep103** adds lines to the file management and case control sections of the bulk data file “*filename_eg.bdf*.” The lines that are added are shown in bold in Appendix E. It also creates the file **prob_prefix.scr**, which contains the 8-character prefix (i.e. *filename*) for use in other file manipulation programs.
- The first NASTRAN run calculates the eigenvectors of the model and writes them to two files for subsequent use as described below. This NASTRAN run utilizes the DMAP alter **dm_103.v** to modify the standard normal modes analysis (solution 103).
 - **egvec.dat** contains the mass-normalized large set of g-size eigenvectors (all degrees of freedom including constrained ones) requested in the modeling phase (see Figure 5). This is an ASCII file with the INPUTT4 DMAP matrix input format [14].
 - **egveH.dat** contains the mass-normalized large set of l-size eigenvectors (all unconstrained degrees of freedom) requested in the modeling phase. This is a binary file.
- **gendisp** – This FORTRAN program generates the displacements fields used in the subsequent solution 106 and 101 runs. The following files are created:
 - **egveCa.dat** contains the mass-normalized selected set of g-size eigenvectors specified in the file **fixed.mod**. This data is a subset of **egvec.dat**. This is an ASCII file with the INPUTT4 DMAP matrix input format.
 - **displ.pr2** contains the number of unique combinations of two modes (IN) and the combinations of those modes used to prescribe the displacement fields imposed for the inverse problems. This is an ASCII file in free format.
 - **displ.pr3** contains the number of unique combinations of three modes (IC) and the combinations of those modes used to prescribe the displacement fields imposed for the inverse problems. This is an ASCII file in free format.
 - **displ.inn** are the prescribed displacement fields used for the linear inverse problem. This is an ASCII file with the INPUTT4 DMAP matrix input format.
 - **displ.inn** are the prescribed displacement fields used for the nonlinear inverse problem. This is an ASCII file with the INPUTT4 DMAP matrix input format.
- **prep106** – This FORTRAN program reads the bulk data file “*filename_st.bdf*” and rewrites it to the file **fixed106_st.bdf**. The program **prep106** adds lines to the file management and case control sections of the bulk data file “*filename_st.bdf*.” In particular, it writes the number of sub-cases (i.e. the number of prescribed displacement fields) required in the subsequent NASTRAN run. The number of sub-cases depends on the number of modes selected (NMOD), number of unique combinations of two modes (IN) from the number of modes selected, and the number of

unique combinations of three modes (IC) from the number of modes selected. It is calculated using the following relation

$$\text{NSUBC} = 2 \times \text{NMOD} + 3 \times \text{IN} + \text{IC}$$

Thus for NMOD = 3, IN = 3 (combinations of modes 1&2, 2&3, and 1&3), IC = 1 (combination of modes 1&2&3) and the number of sub-cases NSUBC = 16. As an example, a listing of the changed lines of file **fixed106_st.bdf** are shown in bold in Appendix F for NMOD = 2.

- The second NASTRAN run calculates the force vector corresponding to the prescribed displacement fields (provided in the file **displ.inn**) in a nonlinear analysis. Because it is not possible to separate the linear and nonlinear components, the resulting force vector contains the combined force. The resulting force vector is written to **f_N.frc**, an ASCII file with the INPUTT4 DMAP matrix input format. The DMAP alter **dm_106.v** is used to modify the standard nonlinear static analysis (solution 106).
- **prep101** – This FORTRAN program reads the bulk data file “*filename_st.bdf*” and rewrites it to the file **fixed101_st.bdf**. The program **prep101** adds lines to the file management and case control sections of the bulk data file “*filename_st.bdf*.” In particular, it writes the number of sub-cases (i.e. the number of prescribed displacement fields) required in the subsequent NASTRAN run. The number of sub-cases is equal to the number of modes selected (NMOD). As an example, a listing of the changed lines of file **fixed101_st.bdf** are shown in bold in Appendix G for NMOD = 2.
- The third NASTRAN run calculates the force vector corresponding to the prescribed displacement fields (provided in the file **displ.inn**) in linear analysis. The resulting force vector is written to **f1_L.frc**, an ASCII file with the INPUTT4 DMAP matrix input format. The DMAP alter **dm_101.v** is used to modify the standard linear static analysis (solution 101).
- **nlcfs** – This FORTRAN program calculates the nonlinear stiffness coefficients of the structure and stores them in the file **nlcfs.dat**. Note that both quadratic a_{jk} and cubic b_{jkl} stiffness coefficients [10-12] are computed in **nlcfs**, but only the cubic terms are stored in **nlcfs.dat** because a zero mean response is assumed. The file **nlcfs.dat** is an ASCII file with the FORTRAN format (1P,3E23.16).

The nonlinear stiffness coefficients computed through this series of operations depend only on the structure (geometry, material properties, and boundary conditions) and not on the loading level or distribution. Therefore, this portion of the analysis only needs to be performed once for a given structural configuration. The effect of loading is addressed in the next sequence of analyses (see Section 3.3).

In order to check the functioning of the entire analysis procedure, it is recommended that a linear analysis be run from this point forward using the two remaining sequences of analyses (see Sections 3.3 and 5) as is. To do so, all nonlinear stiffness coefficients in **nlcfs.dat** should be set to very small (non-zero) values (e.g. 10^{-15}), and the weighting α and β in the file **fixed.par** should be set to 1 and 0, respectively. If everything is working well, the results obtained will be virtually identical to those from a linear solution 111 analysis.

The UNIX script file **doit1.bat** (see Appendix A) runs the first part of the analysis as outlined above in an automated fashion. Some commands in **doit1.bat** may need to be modified depending on the operating system and how NASTRAN is installed. Removal (or transfer to another directory) of previously created ELSTEP output files is necessary for proper program execution. Removal of previously created standard NASTRAN files, e.g. *.f06, *.f04, *.op2, *.log, and *.pch, is optional, but helps to reduced clutter.

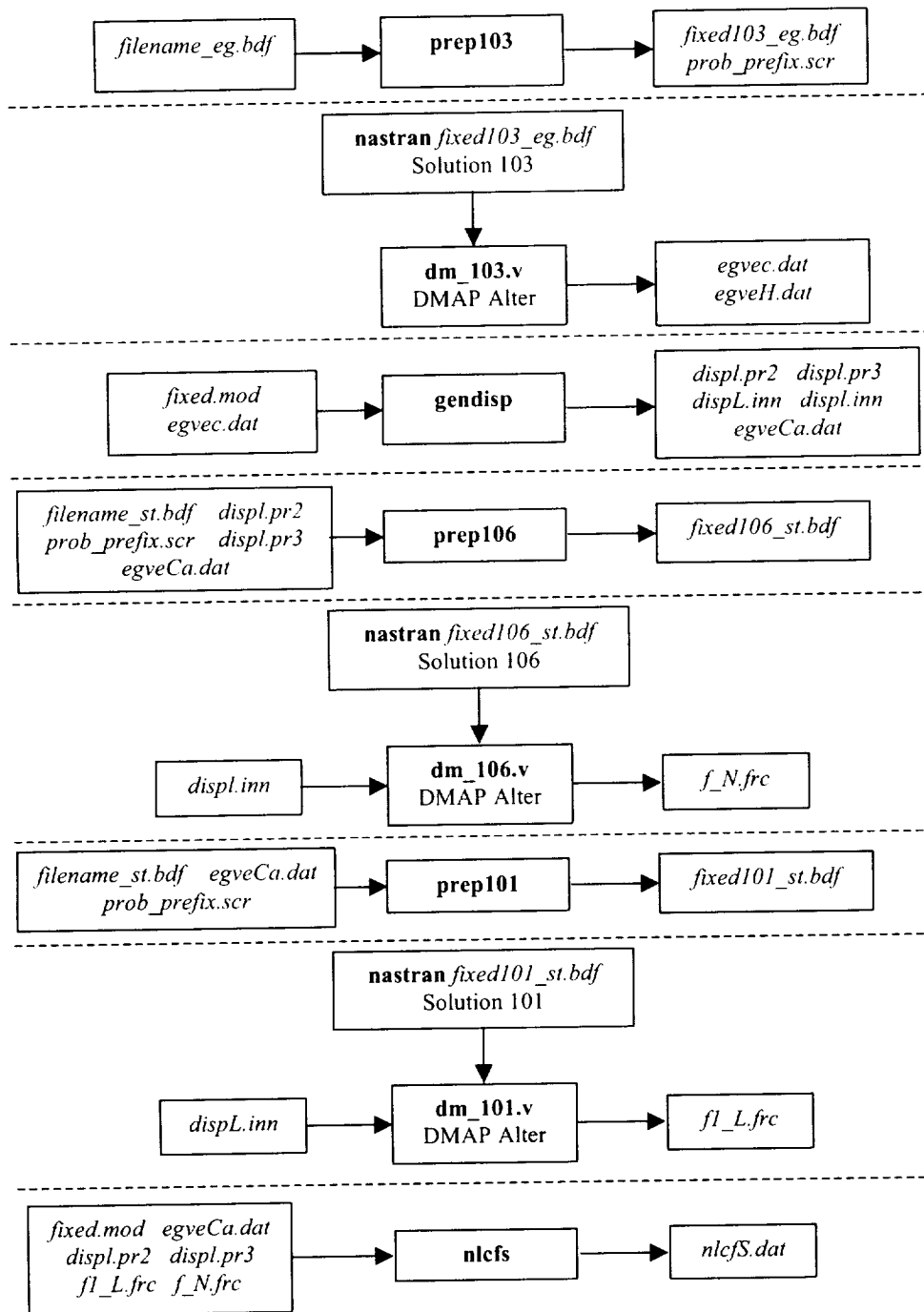


Figure 13: Flow-chart for calculation of nonlinear stiffness coefficients in the FORTRAN implementation.

3.3. Equivalent Linearization Procedure

Both error minimization approaches, force and strain energy, follow the same sequence of analyses. A flowchart is shown in Figure 14. Each operation is separated by a dashed line and is run in the sequence shown from top to bottom. The various operations are described below.

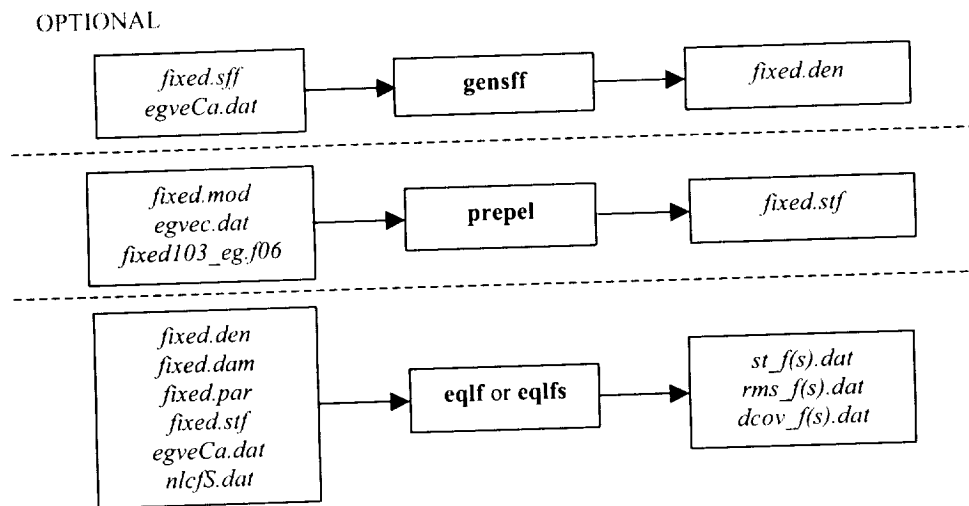


Figure 14: Flow chart for calculation of total equivalent stiffness matrix and RMS displacements in the FORTRAN implementation

- **gensff** – The first operation is optional and is used to develop the frequency dependent spectral density matrix of the loading in modal coordinates. Ideally, the spectral density matrix of the loading in physical degrees of freedom could be obtained through a DMAP alter of solution 111, then pre- and post-multiplied by the selected set of eigenvectors to obtain the modal spectral density matrix. In this way, the usual manner of computing nodal loads is retained, i.e. loads are determined internally within NASTRAN using information provided in the bulk data file written by PATRAN. Unfortunately, efforts to extract this information have not been successful. Instead, the utility program **gensff** may be used to compute the modal spectral density matrix for a couple of special cases – a uniformly distributed acoustic pressure (which may also be used to specify an inertial load) and point forces and moments. In each case, the loadings are fully correlated giving a real-valued spectral density matrix in physical coordinates. The mass-normalized selected set of g-size eigenvectors are read from **egveCa.dat** and information about the loading is read from **fixed.sff**. The computed modal spectral density matrix is written to **fixed.den** (as described in Section 3.1.2).

- The parameter file **fixed.sff** specifies the following information in free format:

LINE 1: Load option (LOADOPT)
 1 = uniformly distributed pressure with uniform mesh
 2 = point force(s) and moment(s)

If LOADOPT = 1

LINE 2: AREA, NOD, NODB

AREA – Surface area of structure

NOD – Total number of nodes in the model

NODB – Number of constrained (boundary) nodes

Forces at the grid points are obtained by multiplying the pressure level by the area and distributing these over the mesh, hence the need for a uniform mesh. If a non-uniform mesh is used, a pressure loading may be specified using $\text{LOADOPT} = 2$ with forces input at each grid point.

LINE 3: NPT

NPT – Number of breakpoints in the PSD ($2 \leq \text{NPT} \leq 50$)

LINE 4: (repeated NPT times): FREQ, PSD

FREQ, PSD – Breakpoint pairs for PSD definition of uniformly distributed pressure. Values of PSD will be linearly interpolated between breakpoint pairs at the frequency increment specified in **fixed.par**. FREQ is the frequency specified in Hz. PSD is the single-sided pressure PSD in units^2/Hz and is the same as would be used on the TABRND1 card for a solution 111 analysis. Note that **gensff** converts the load internally to a double-sided value in $\text{units}^2/\text{rad/sec}$.

Note that in this case of a uniformly distributed pressure on a uniformly distributed mesh, the forces for each grid point are identical and act in the z-direction. Hence, all non-zero auto- and cross-spectral density values in S_{ff} are equal. Further, each column is identical. These properties speed the computation of \tilde{S}_{ff} .

If $\text{LOADOPT} = 2$

LINE 2: NLOAD

NLOAD – Number of fully correlated point forces and moments.

LINE 3: NPT

NPT – Number of breakpoints in the PSD ($2 \leq \text{NPT} \leq 50$)

(Lines 4-5 repeated NLOAD times)

LINE 4: NODF, DOF1-DOF6

NODF – Grid point at which point force is applied

DOF1-DOF6 – Multiplier for each of six degree-of-freedoms at each grid point. An arbitrarily oriented force or moment may be applied in this fashion.

LINE 5: (repeated NPT times): FREQ, PSDF

FREQ, PSDF – Breakpoint pairs for PSD definition of point forces. Values of PSDF will be linearly interpolated between breakpoint pairs at the frequency increment specified in **fixed.par**. FREQ is the frequency in Hz. PSDF is the single-sided force PSD in units^2/Hz and is the same as would be used on the TABRND1 card for a solution 111 analysis. Note that **gensff** converts the load internally to a double-sided value in $\text{units}^2/\text{rad/sec}$.

Note that in this case, the forces for each grid point are not identical. Therefore, the non-zero auto- and cross-spectral density values in S_{ff} are unequal and each column is not identical. These properties make the computation of \tilde{S}_{ff} more intensive.

Note that in general, the cross-spectral densities of the input loading are required to specify S_{ff} . However, since both load options require fully correlated (in phase) loadings, the cross-spectral density (CSD) terms may be obtained from the auto-spectral density (PSD) from the following relationship:

$$CSD_{ij} = \pm \sqrt{(PSD_i)(PSD_j)}$$

This allows for the specification of only the PSD values, and not both the PSD and CSD values, as indicated above. Note that **gensff** as currently implemented only takes the positive square root. Therefore, the specified loadings must have the same direction. Failure to do so will result in an inconsistency in the sign of the diagonal and off-diagonal terms in S_{ff} .

In the upcoming examples, the following may be helpful. The values specified above for the spectral density matrix of the loading in physical coordinates (S_{ff}) may be determined from an RMS level for a particular bandwidth using the following relation

$$S_{ff} = \frac{RMS^2/BW}{2 \times 2\pi}$$

where BW is the frequency bandwidth in Hz. For example, for an RMS acoustic pressure of 2048 Pa (160.21 dB) and 0-1024 Hz bandwidth, the double-sided value of S_{ff} is 325.95 Pa²/rad/sec. The corresponding single-sided value, which should be used in **fixed.sff**, is 4096 Pa²/Hz.

The following is an example of **fixed.sff** for the uniformly distributed pressure indicated above:

```
1
0.0903224 609 96
0.      4096.
1024.   4096.
```

The following is an example of **fixed.sff** for point forces. The force at grid point 305 acts in the z-direction and linearly increases in magnitude from 0 to 5 N²/Hz over the bandwidth 0-512 Hz. A second force acts in the x-direction at grid point 220 with a constant magnitude of 2 N²/Hz over the frequency range of 0-512 Hz.

```
2
2
2
305      0.0      0.0      1.0      0.0      0.0      0.0
0.      0.0
512.     5.0
220      2.0      0.0      0.0      0.0      0.0      0.0
0.      1.0
```

512. 1.0

Note that when converting from a uniform pressure load specified by load option 1 to a force load specified by load option 2, the PSD values specified in option 1 must be multiplied by the area squared.

- **prepel** – This FORTRAN program reads the NASTRAN f06 file **fixed103_eg.f06** produced in the second step of the previous procedure (see Section 3.2) and writes the selected natural frequencies to the file **fixed.stf**. This is an ASCII file in free format.
- The third operation performs the equivalent linearization procedure through the FORTRAN programs **eq1f** and **eq1s**, for the force error and strain energy error minimization approaches, respectively. Each of these alters produce the following output:
 - **st_f.dat** and **st_s.dat** contain the total equivalent stiffness matrix in modal coordinates for the force and strain energy error minimization approaches, respectively. These are ASCII files with the INPUTT4 DMAP matrix input format.
 - **rms_f.dat** and **rms_s.dat** contain the root-mean-square displacements in physical coordinates for all degrees of freedom. These are ASCII files in free format.
 - **dcov_f.dat** and **dcov_s.dat** contain the cross-covariance matrix of modal displacements. These are ASCII files in free format.

The UNIX script files **doit2_f.bat** and **doit2_s.bat** (see Appendix A) run the second part of the analysis as outlined above in an automated fashion.

3.3.1. Convergence

The weightings α and β , and the convergence criteria ϵ , play a significant role in the convergence of the equivalent linearization procedure. Because the system is nonlinear, it is possible for multiple solutions to exist for a given loading condition. The selection of different weightings may cause the equivalent linearization procedure to converge to different solutions. Therefore, it is advisable to explore the solution space by trying out different weightings before selection of a converged solution. If the root-mean-square displacements from the equivalent linearization procedure, as reported in **rms_f.dat** or **rms_s.dat**, differ with the choice of weighting, then further analyses may be warranted, such as observation of the deflected shape or the equivalent linear power spectral density from a post-processing analysis (see Section 5).

4. DMAP Implementation

As in the FORTRAN implementation, the analysis for the DMAP implementation is subdivided into two parts. The first part calculates the nonlinear stiffness coefficients. The second part performs one of two equivalent linearization procedures to compute the total equivalent stiffness matrix and RMS displacements. A description of optional post-processing is provided in Section 5.

Note that since some files bear the same name but have a different format from those in the FORTRAN implementation, both FORTRAN and DMAP implementations should not be run from the same directory.

4.1. Input Files

In addition to the bulk data files discussed in Section 2, several additional input files are required to specify various parameters used in the analysis. These are detailed below.

4.1.1. Modes Selection File

As in the FORTRAN implementation, the file **fixeD.mod** indicates which eigenvectors out of the large set of eigenvectors participate in the analysis. The format for **fixeD.mod** is a INPUTT4 DMAP matrix input format as specified below.

- LINE 1: Number of columns (always 2), number of selected modes (NMOD), matrix form (always 2 for rectangular), type of matrix (always 2 for real, double precision), name of the matrix (always PHG), FORTRAN format specification (always 1P,3E23.16).
- LINE 2: Column number (always 1), row position of first nonzero term (always 1), number of real double-precision entries on the following line (NMOD).
- LINE 3: Selected modes (up to three per line). Note that these numbers are input as real and converted to integers within the program
- LINE 4: Column number (always 2), row position of first nonzero term (always 1), number of real double-precision entries on the following line (NMOD).
- LINE 5: Scaling coefficients for each eigenvector selected (up to three per line). See Section 3.1.1 for a discussion of these terms.

A sample listing of **fixeD.mod** with two selected modes (NMOD=2) is provided below.

```
2      2      2      2PHG      1P,3E23.16
1      1      2
1.0000000000000000E+00 4.0000000000000000E+00
2      1      2
1.0000000000000000E-04 1.0000000000000000E-04
```

It is clear that the size of this file depends on the number of modes selected. Since the format can be somewhat confusing, a sample listing with four selected modes (NMOD=4) is also provided below.

```
2      4      2      2PHG      1P,3E23.16
1      1      4
1.0000000000000000E+00 3.0000000000000000E+00 7.0000000000000000E+00
1.0000000000000000E+01
```

```

      2      1      4
1.0000000000000000E-04 1.0000000000000000E-04 1.0000000000000000E-04
1.0000000000000000E-04

```

4.1.2. Spectral Density Loads File

The file **fixeD.den** is analogous to **fixed.den** in the FORTRAN implementation, except that the spectral density matrix of the applied loads in modal coordinates \tilde{S}_{ff} is independent of frequency in the DMAP implementation, i.e. only flat spectra may be applied. The format for **fixeD.den** is a INPUTT4 DMAP matrix input format as specified below.

LINE 1: Number of selected modes (NMOD), number of selected modes (NMOD), matrix form (always 1 for square), type of matrix (always 4 for complex, double precision), name of the matrix (always PHG), FORTRAN format specification (always 1P,6E13.6).

LINE 2: Column number (always 1), row position of first nonzero term (always 1), number of real double-precision entries on the following line (2 x NMOD).

LINE 3: First column of modal spectral density matrix (NMOD entries, up to 3 complex entries per line). Each entry is complex with the imaginary part equal to zero.

(Lines 2 and 3 repeat for NMOD selected modes)

2ND TO LAST LINE:

Column number (always NMOD), row position of first nonzero term (always 1), number of real double-precision entries on the following line (2 x NMOD).

LAST LINE:

Last column of modal spectral density matrix (NMOD entries, up to 3 complex entries per line).

A sample listing of **fixeD.den** for two selected modes is provided below.

```

      2      2      1      4PHG      1P,6E13.6
      1      1      4
0.683899E+01 0.000000E+00 0.227952E+01 0.000000E+00
      2      1      4
0.227952E+01 0.000000E+00 0.759793E+00 0.000000E+00

```

To be explicit, the above specifies the following frequency-independent spectral density matrix of applied loads in modal coordinates:

$$\tilde{S}_{ff} = \begin{bmatrix} \tilde{S}_{ff11} & \tilde{S}_{ff12} \\ \tilde{S}_{ff21} & \tilde{S}_{ff22} \end{bmatrix} = \begin{bmatrix} (6.84,0) & (2.28,0) \\ (2.28,0) & (0.76,0) \end{bmatrix}$$

This matrix is computed as

$$\tilde{S}_{ff} = \phi^T S_{ff} \phi \quad (2)$$

where S_{ff} is the real-valued spectral density matrix of the load in physical coordinates (double-sided spectrum level in units²/rad/sec) and ϕ is the subset of selected eigenvectors. The eigenvectors to be used in equation (2) are the set of selected g-size (all degrees of freedom including constrained ones)

eigenvectors from the file **egveCa.dat** (see Section 4.2). As in the FORTRAN implementation, only correlated inputs are permitted. Values of S_{ff} must be converted to single-sided values as indicated in Section 3.1.2 for use in a solution 111 post-processing analysis.

In order to facilitate generation of the modal spectral density matrix, the utility program **gensffD** (described in Section 4.3) may be used to automatically compute the values and write them to **fixeD.den** for a limited set of loading conditions.

4.1.3. Damping Matrix File

The file **fixeD.dam** is analogous to **fixed.dam** in the FORTRAN implementation. It contains the diagonal modal damping matrix. The format for **fixeD.dam** is a INPUTT4 DMAP matrix input format as specified below.

LINE 1: Number of selected modes (NMOD), number of selected modes (NMOD), matrix form (always 1 for square), type of matrix (always 2 for real, double precision), name of the matrix (always PHG), FORTRAN format specification (always 1P,3E23.16).

LINE 2: Row and column position of first modal damping values (always 1, 1), number of real double-precision entries on the following line (always 1).

LINE 3: First modal damping value

(Lines 2 and 3 repeat for NMOD selected modes)

2ND TO LAST LINE:

Row and column position of last modal damping values (always NMOD, NMOD), number of real double-precision entries on the following line (always 1).

LAST LINE:

Last modal damping value.

A sample listing of **fixeD.dam** with two selected modes is provided below.

```

      2      2      1      2PHG      1P,3E23.16
      1      1      1
0.14661800000000000E+02
      2      2      1
0.14661800000000000E+02
```

In order to specify the same damping for a solution 111 post-processing analysis, the values must be converted to percent critical damping as discussed in Section 3.1.3.

4.1.4. Parameter File

The file **fixeD.par** is analogous to **fixed.par** in the FORTRAN implementation. It contains various additional parameters required for the analysis. The format for **fixeD.par** is a INPUTT4 DMAP matrix input format as specified below.

LINE 1: Number of columns (always 1), number of rows (always 6), matrix form (always 2 for rectangular), type of matrix (always 2 for real, double precision), name of the matrix (always PHG), FORTRAN format specification (always 1P,3E23.16).

LINE 2: Column number (always 1), row position of first nonzero term (always 1), number of real double-precision entries on the following line (always 6).

LINE 3: Minimum and maximum frequency range of the analysis (Hz), frequency increment used in the analysis (Hz). Note that the maximum frequency range should be selected well beyond the frequency of the highest selected mode. This is necessary so that the resonant frequencies don't shift out of the analysis bandwidth as nonlinearity increases.

LINE 4: Weightings α and β , and convergence criteria ε (see Section 3.1.4 for description). Note that these values are put on a fourth line because the format on line 1 indicates up to three values per line. The choice of weightings may influence the solution to which the system converges, as discussed in Section 3.3.1.

A sample listing of **fixeD.par** is provided below.

```

1          6          2          2PHG          1P, 3E23.16
1          1          6
0.0000000000000000E+00 1.0240000000000000E+03 0.2500000000000000E+00
0.4000000000000000E+00 0.6000000000000000E+00 0.5000000000000000E-02

```

4.2. Calculation of Nonlinear Stiffness Coefficients

The method of calculating the nonlinear stiffness coefficients is shown in the flowchart in Figure 15. Each operation is separated by a dashed line and is run in the sequence shown from top to bottom. The various operations are described below.

- **prep103D** – This FORTRAN program is the analogue to **prep103** in the FORTRAN implementation. It reads the bulk data file “*filename_eg.bdf*” and rewrites it to the file **fixeD103_eg.bdf**. The program **prep103D** adds lines to the file management and case control sections of the bulk data file “*filename_eg.bdf*.” The lines that are added are shown in bold in Appendix H. It also creates the file **prob_prefix.scr**, which contains the 8-character prefix (i.e. *filename*) for use in other file manipulation programs.
- The first NASTRAN run is analogous to the FORTRAN implementation, except that the DMAP alter **dm_103D.v** also generates the displacement fields. These were generated by **gendisp** in the FORTRAN implementation. The following files are created:
 - **egvec.dat** contains the mass-normalized large set of g-size eigenvectors (all degrees of freedom including constrained ones) requested in the modeling phase (see Figure 5). It contains the same information as that produced in the FORTRAN implementation, but in a binary format.
 - **egveH.dat** contains the mass-normalized large set of l-size eigenvectors (all unconstrained degrees of freedom) requested in the modeling phase. This binary file is identical to that produced in the FORTRAN implementation.
 - **egveCa.dat** contains the mass-normalized selected set of g-size eigenvectors specified in the file **fixeD.mod**. It is identical to that produced in the FORTRAN implementation.
 - **egveC.dat** is a binary version of **egveCa.dat**. There is no analogue in the FORTRAN implementation.
 - **displ.pr2** contains the number of unique combinations of two modes (IN) and the combinations of those modes used to prescribe the displacement fields imposed for the inverse problems. It contains the same information as that produced in the FORTRAN implementation, but in the INPUTT4 DMAP matrix input format.

- **displ.pr3** contains the number of unique combinations of three modes (IC) and the combinations of those modes used to prescribe the displacement fields imposed for the inverse problems. It contains the same information as that produced in the FORTRAN implementation, but in the INPUTT4 DMAP matrix input format.
- **displ.pr4** contains additional information used to prescribe the displacement fields imposed for the inverse problems. This is an ASCII file with the INPUTT4 DMAP matrix input format. There is no analogue in the FORTRAN implementation.
- **displ.L.inn** are the prescribed displacement fields used for the linear inverse problem. It contains the same information as that produced in the FORTRAN implementation, but in a binary format.
- **displ.inn** are the prescribed displacement fields used for the nonlinear inverse problem. It contains the same information as that produced in the FORTRAN implementation, but in a binary format.
- **prep106D** – This FORTRAN program is the analogue to **prep106** in the FORTRAN implementation. It reads the bulk data file “*filename_st.bdf*” and rewrites it to the file **fixeD106_st.bdf**. The program **prep106D** adds lines to the file management and case control sections of the bulk data file “*filename_st.bdf*.” As an example, a listing of the changed lines of file **fixeD106_st.bdf** are shown in bold in Appendix I for NMOD = 2.
- The second NASTRAN run is analogous to the FORTRAN implementation. In the DMAP implementation, the file **f_N.frc** is binary. The DMAP alter **dm_106D.v** is used to modify the standard nonlinear static analysis (solution 106).
- **prep101D** – This FORTRAN program is the analogue to **prep101** in the FORTRAN implementation. It reads the bulk data file “*filename_st.bdf*” and rewrites it to the file **fixeD101_st.bdf**. The program **prep101D** adds lines to the file management and case control sections of the bulk data file “*filename_st.bdf*.” As an example, a listing of the changed lines of file **fixeD101_st.bdf** are shown in bold in Appendix J for NMOD = 2.
- The third NASTRAN run is analogous to the FORTRAN implementation. In the DMAP implementation, the file **f1_L.frc** is binary. The DMAP alter **dm_101D.v** is used to modify the standard linear static analysis (solution 101).
- The fourth NASTRAN run is the DMAP analogue of the FORTRAN program **nlcfs**. The file **nlcfs.bdf** is a standalone DMAP alter, not a modification of a standard NASTRAN solution. Like **nlcfs**, both quadratic a_{jk} and cubic b_{jkl} stiffness coefficients are computed in **nlcfs.bdf**, but only the cubic terms are stored in **nlcfS.dat** because a zero mean response is assumed. The nonlinear stiffness coefficients are also stored in the ASCII file **nlcfS.dat**, but in the INPUTT4 DMAP matrix input format.

As in the FORTRAN implementation, this portion of the analysis only needs to be performed once for a given structural configuration. Similarly, proper functioning of the entire analysis procedure should be checked using the method indicated in Section 3.2.

The UNIX script file **doitD1.bat** (see Appendix A) runs the first part of the analysis as outlined above in an automated fashion.

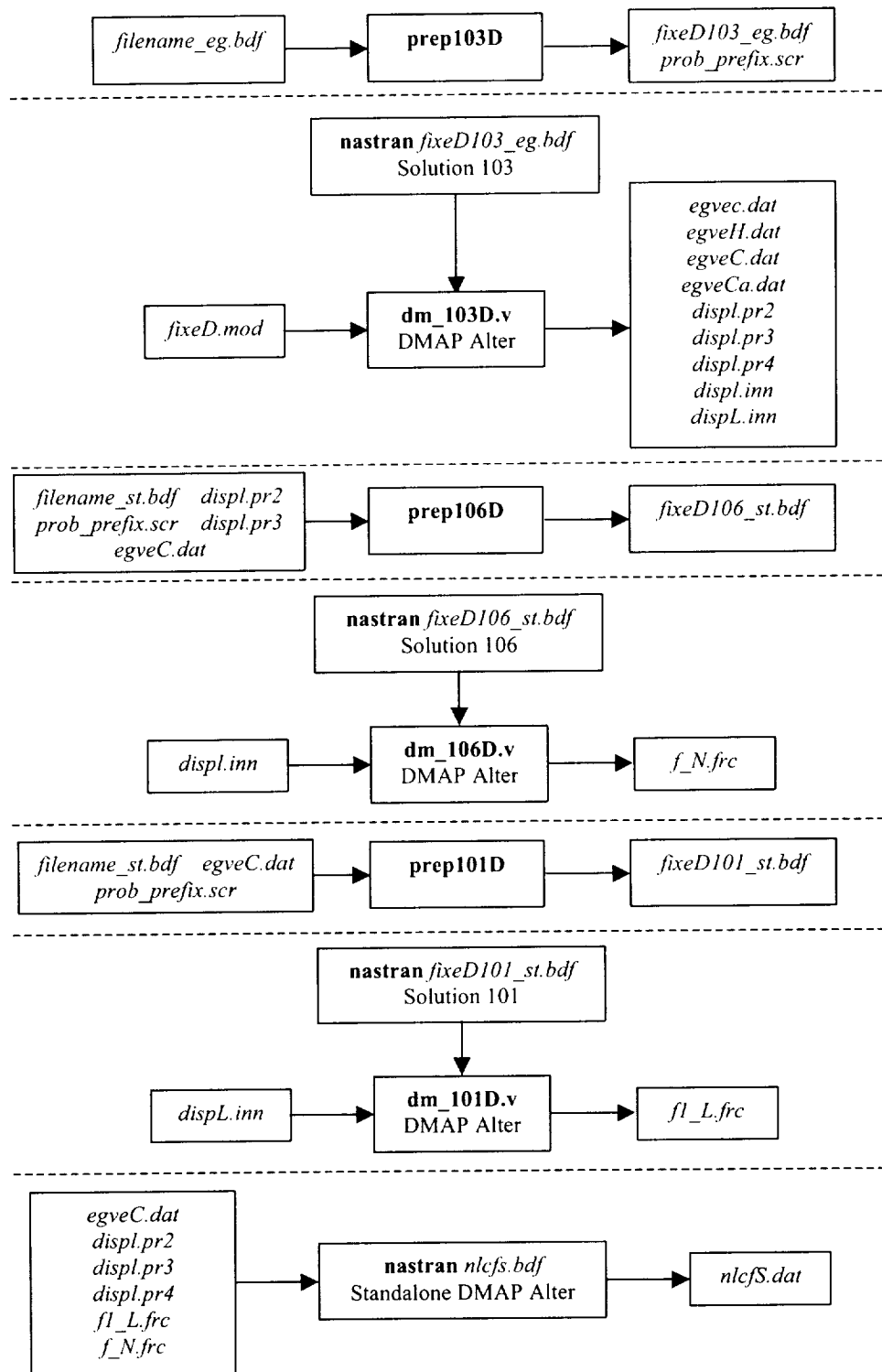


Figure 15: Flow-chart for calculation of nonlinear stiffness coefficients in the DMAP implementation.

4.3. Equivalent Linearization Procedure

Like the FORTRAN implementation, both force and strain energy error minimization approaches follow the same sequence of analyses, as shown in Figure 16. Each operation is separated by a dashed line and is run in the sequence shown from top to bottom. The various operations are described below.

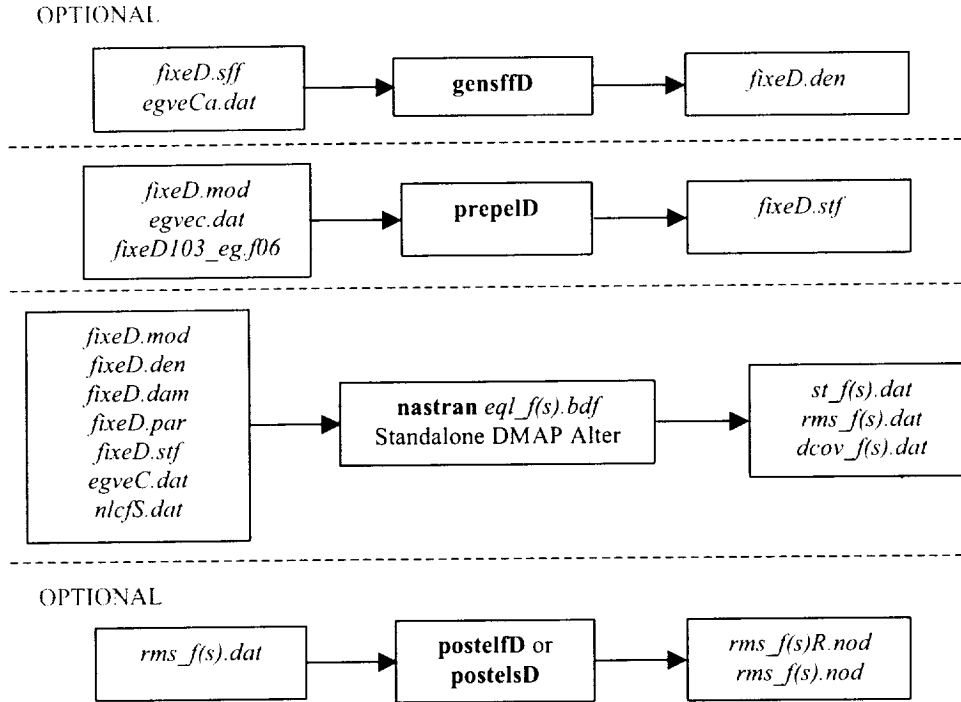


Figure 16: Flow chart for calculation of total equivalent stiffness matrix and RMS displacements in the DMAP implementation.

- gensffD** – The first operation is optional and is used to develop the spectral density matrix of the loading in modal coordinates. It is analogous to the program **gensff** in the FORTRAN implementation, except that in the DMAP implementation, the spectral density matrices are independent of frequency, i.e. only flat spectra may be specified. The program **gensffD** may be used to compute the modal spectral density matrix for a couple of special cases – a uniformly distributed acoustic pressure (which may also be used to specify an inertial load) and point forces and moments. In each case, the loadings are fully correlated giving a real-valued spectral density matrix in physical coordinates. The mass-normalized selected set of g-size eigenvectors are read from **egveCa.dat** and information about the loading is read from **fixeD.sff**. The computed modal spectral density matrix is written to **fixeD.den** (as described in Section 4.1.2).
 - The parameter file **fixeD.sff** specifies the following information in free format:

LINE 1: Load option (LOADOPT)

1 = uniformly distributed pressure with uniform mesh

2 = point force(s) and moment(s)

If LOADOPT = 1

LINE 2: PSD, AREA, NOD, NODB

PSD – Spectrum level of uniformly distributed pressure in units²/Hz. This level is a single-sided value and is the same as would be used on the TABRND1 card for sol 111 analysis. Note that **gensffD** converts the load internally to a double-sided value in units²/rad/sec.

AREA – Surface area of structure

NOD – Total number of nodes in the model

NODB – Number of constrained (boundary) nodes

Forces at the grid points are obtained by multiplying the pressure level by the area and distributing these over the mesh, hence the need for a uniform mesh. If a non-uniform mesh is used, a pressure loading may be specified using LOADOPT = 2 with forces input at each grid point.

Note that in this case of a uniformly distributed pressure on a uniformly distributed mesh, the forces for each grid point are identical and act in the z-direction. Hence, all non-zero auto- and cross-spectral density values in S_{ff} are equal. Further, each column is identical. These properties speed the computation of \tilde{S}_{ff} .

If LOADOPT = 2

LINE 2: NLOAD

NLOAD – Number of fully correlated point forces and moments.

LINE 3 (repeated NLOAD times): NODF, PSDF, DOF1-DOF6

NODF – Grid point at which point force is applied

PSDF – Spectrum level of force in units²/Hz. This level is a single-sided value and is the same as would be used on the TABRND1 card for sol 111 analysis. Note that **gensffD** converts the load internally to a double-sided value in units²/rad/sec.

DOF1-DOF6 – Multiplier for each of six degree-of-freedom at each grid point. An arbitrarily oriented force or moment may be applied in this fashion.

Note that in this case, the forces for each grid point are not identical. Therefore, the non-zero auto- and cross-spectral density values in S_{ff} are unequal and each column is not identical. These properties make the computation of \tilde{S}_{ff} more intensive.

Note that in general, the cross-spectral densities of the input loading are required to specify S_{ff} . However, since both load options require fully correlated (in phase) loadings, the cross-spectral density (CSD) terms may be obtained from the auto-spectral density (PSD) from the following relationship:

$$CSD_{ij} = \pm \sqrt{(PSD_i)(PSD_j)}$$

This allows for the specification of only the PSD values, and not both the PSD and CSD values, as indicated above. Note that **gensffD** as currently implemented only takes the positive square root. Therefore, the specified loadings must have the same direction.

Failure to do so will result in an inconsistency in the sign of the diagonal and off-diagonal terms in S_{ff} .

The following is an example of a uniformly distributed pressure of 4096 Pa²/Hz:

```
1
4096 0.0903224 609 96
```

The following is an example of a point force of 5 N²/Hz in the z-direction at grid point 305 and a point force of 2 N²/Hz in the x-direction at grid point 220:

```
2
2
305      5.0      0.0      0.0      1.0      0.0      0.0      0.0
220      1.0      2.0      0.0      0.0      0.0      0.0      0.0
```

- **prepelD** – This FORTRAN program is the analogue to **prepel** in the FORTRAN implementation. It reads the NASTRAN f06 file **fixeD103_eg.f06** produced in the second step of the previous procedure (see Section 4.2) and writes the selected natural frequencies to the file **fixeD.stf**. Here, **fixeD.stf** is an ASCII file with the INPUTT4 DMAP matrix input format.
- The third operation is the DMAP analogue of the FORTRAN programs **eqlf** and **eqls**. This NASTRAN run performs the equivalent linearization procedure through the standalone DMAP alters **eqf.f.bdf** and **eqs.f.bdf**, for the force error and strain energy error minimization approaches, respectively. Each of these alters produce the following output:
 - **st.f.dat** and **st.s.dat** contain the total equivalent stiffness matrix in modal coordinates for the force and strain energy error minimization approaches, respectively. These are ASCII files with the INPUTT4 DMAP matrix input format.
 - **rms.f.dat** and **rms.s.dat** contain the root-mean-square displacements in physical coordinates for all degrees of freedom. These are ASCII files with the INPUTT4 DMAP matrix input format.
 - **dcov.f.dat** and **dcov.s.dat** contain the cross-covariance matrix of modal displacements. These are ASCII files with the INPUTT4 DMAP matrix input format.
- The fourth (optional) operation has no FORTRAN analogue. It should be performed only if graphical display of results in PATRAN is desired. FORTRAN programs **postelfD** and **postelsD** read the files **rms.f.dat** and **rms.s.dat**, respectively, and separate the translational and rotational degrees of freedom. The RMS translational displacements for all grid points are written to either **rms.f.nod** or **rms.s.nod**, depending on the error minimization procedure used. The RMS rotational displacements for all grid points are written to either **rms.fR.nod** or **rms.sR.nod**, depending on the error minimization procedure used. These ASCII results files may be imported from within PATRAN for plotting, etc.

There are a couple of comments worth noting about the analysis described in this section. The first has to do with creation of the files **eqf.f.f04** and **eqs.f.f04** when running **eqf.f.bdf** and **eqs.f.bdf**, respectively. These files can be very large due to the amount of information written by NASTRAN. Unfortunately, a means of suppressing creation of these files or minimizing the information written to them has not been identified. Although the files can be deleted immediately following successful execution of the run, ample disk space must be available to successfully complete the run. This problem is most acute when running the strain energy error minimization analysis, as the number of iterations for convergence at highly nonlinear levels can be in the hundreds or thousands (compared with ten or less typical of the force

error minimization approach). Consequently, large temporary disk space is recommended when running the strain energy error minimization analysis.

The second item worth mentioning relates to the calculation of RMS displacements in physical coordinates when running **eq1.f.bdf** and **eq1.s.bdf**. Following calculation of the total equivalent stiffness matrix and cross-covariance matrix in modal coordinates, the RMS displacements in physical coordinates are recovered and stored in the files **rms.f.dat** and **rms.s.dat**. This process can take a significant amount of time as presently implemented due to the amount of disk I/O that occurs, especially for large models. If the intention is to perform a post-processing analysis to determine displacement PSDs and/or stress/strain information, as outlined in Section 5, a more efficient means of getting RMS displacements in physical coordinates is to request this information as part of that analysis. In this case, the value of RMSDISP should be changed from 1 to 0 in **eq1.f.bdf** and **eq1.s.bdf** to suppress computation of this data.

The UNIX script files **doitD2.f.bat** and **doitD2.s.bat** (see Appendix A) run the second part of the analysis as outlined above in an automated fashion.

4.3.1. Convergence

The reader should refer to Section 3.3.1 for a discussion on convergence of the equivalent linearization procedure.

5. Post-Processing for Displacement, Stress and Strain PSDs

An optional post-processing procedure may be performed to compute RMS displacements, displacement PSDs, RMS stress/strain, and stress/strain PSDs. It applies to both FORTRAN and DMAP implementations. A flowchart of the implementation is shown in Figure 17. Each operation is separated by a dashed line and is run in the sequence shown from top to bottom. The various operations are described below.

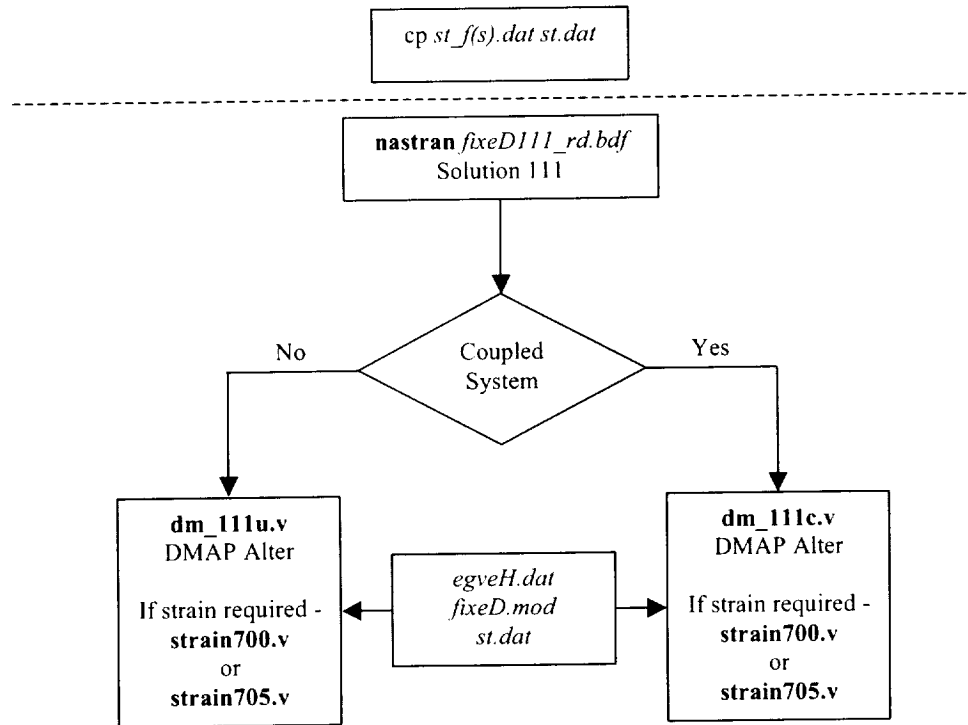


Figure 17: Flow chart for post-processing.

- The first operation simply copies the total equivalent stiffness matrix from either the force or strain energy error minimization approaches to the file **st.dat**. This allows stiffness matrices from both approaches to coexist in the same directory.
- The second operation performs a post-processing analysis using the total equivalent stiffness matrix, obtained from the equivalent linearization analysis, in place of the linear stiffness matrix in a NASTRAN random analysis (solution 111).

Note that since the total equivalent stiffness matrix in modal coordinates is symmetric, but not (in general) diagonal, the modal system of equations to be solved is coupled. In order to reduce the computational effort, the option is provided to analyze an uncoupled system, which is formed through an additional transformation process. The uncoupled system has a different total equivalent stiffness matrix (now diagonal and in transformed coordinates), but utilizes the same diagonal damping matrix and unit mass matrix as the coupled system. Analysis of the uncoupled system is significantly faster than the coupled system, but is not as accurate. It is recommended that the root-mean-square displacements from the post-processing analysis be compared with those obtained from the equivalent linearization analysis. In all circumstances, post-processing results from the coupled system should be comparable to values obtained from the equivalent

linearization analysis. Only small differences, on the order of a few percent, should be found (see Section 6.4 for example), since the loading is specified equivalently but differently between the equivalent linearization and post-processing analyses. Post-processing results from the uncoupled system may or may not be comparable to values obtained from the equivalent linearization analysis. Thus, if results from the uncoupled analysis significantly differ from results from the equivalent linearization analysis, it is recommended that the coupled system be used for post-processing. The choice of coupled versus uncoupled system is made in the file **fixeD111_rd.bdf** (see Appendix D), through selection of the appropriate DMAP alter as shown in Figure 17.

If root-mean-square strain results are to be recovered, the alters **strain700.v** or **strain705.v** should be utilized. For NASTRAN version 70.0, the alter **strain700.v** should be used. For NASTRAN versions 70.5 and 70.7, the alter **strain705.v** should be used. If root-mean-square stress results are to be recovered, neither alter should be utilized. The choice of which alter to use (if any) is made in the file **fixeD111_rd.bdf** (see Appendix D). The file **Strain-Recovery-Notes.pdf** (see Appendix A) documents other commands required in the case control section for the different NASTRAN versions. Note that stress and strain results are recovered using linear strain/displacement relationships. This approach appears to be acceptable for the beam and plate analyses conducted to date.

Because the post-processing analysis is performed on the equivalent linear system, all PSDs appear linear in character. In other words, at nonlinear response levels, the PSDs exhibit neither peak broadening nor the same shifts in resonant frequencies as the truly nonlinear system.

Shifted resonant frequencies of the equivalent linear system are not directly reported since a modified solution 103, using the total equivalent stiffness matrix in place of the linear stiffness matrix, is not presently available. The shifted resonant frequencies may be determined by scanning the punch file **fixeD111_rd.pch** for peaks in the displacement PSD (when flat spectra are applied). Note that it may be necessary to run the post-processing analysis more than once so that the frequency ranges specified on the TABDMP1 card are centered about the shifted peaks of the equivalent linear system. The frequency range for each critical damping value is typically set to bracket the corresponding resonant peak at mid-span. As indicated in Section 3.1.3, the relationship between percent critical damping and modal damping values should remain based on the linear eigenvalues.

The UNIX script files **doitD3_f.bat** and **doitD3_s.bat** (see Appendix A) run the post-processing part of the analysis as outlined above in an automated fashion.

6. Example Problem

A simply support aluminum plate under normal acoustic excitation was chosen to illustrate the execution of ELSTEP and to generate sample results which may be used subsequently to verify proper execution with future NASTRAN and PATRAN releases. Consequently, many results are reported to a large number of significant digits.

The rectangular plate chosen measured 0.254m x 0.3556m x 0.00102m. All sides were simply supported. Material properties were:

$$E = 7.3 \times 10^{10} \text{ Pa}$$

$$\nu = 0.3$$

$$\rho = 2763 \text{ kg/m}^3$$

giving a flexural rigidity of

$$D = \frac{Eh^3}{12(1-\nu^2)} = 7.094 \frac{\text{kg-m}^2}{\text{s}^2}$$

Natural frequencies may be computed from the following relation [15]

$$\omega_{mn} = \sqrt{\frac{D}{\rho} \left[\left(\frac{m\pi}{a} \right)^2 + \left(\frac{n\pi}{b} \right)^2 \right]}$$

giving

$$\omega_{11} = 58.34 \text{ Hz}$$

$$\omega_{13} = 216.01 \text{ Hz}$$

for the first two symmetric modes.

A finite element model of this structure was built in PATRAN having 560 CQUAD4 elements measuring 0.0127m square. The first two symmetric modes (modes 1 and 4) of this model had natural frequencies of 58.38 and 217.27 Hz. These two modes were selected as participating modes in the ELSTEP analysis as per the example files in Sections 3.1.1 and 4.1.1. Modal damping was specified as per the example in Sections 3.1.3 and 4.1.3, giving percent critical damping values of 2% and 0.54% for modes 1 and 4, respectively.

A spatially uniform distributed normal acoustic pressure with RMS value of 2048 Pa, and a single-sided spectrum level of 4096 Pa²/Hz over a 0-1024 Hz bandwidth was applied to the plate. Using the example input file for a uniformly distributed pressure (described in Sections 3.3 and 4.3) for the programs **gensff** and **gensffd**, the modal spectral density matrix generated is as shown in Sections 3.1.2 and 4.1.2. Note that this load level gives rise to a highly nonlinear response.

6.1. Nonlinear Stiffness Results

The cubic nonlinear stiffness coefficients were computed and written to **nlcfS.dat**. For the FORTRAN implementation, **nlcfS.dat** is written as:

```
4.1355686358055371E+12-1.6827938202459033E+12
2.3630088637711590E+13
4.0490613582441501E+07
```

```

-5.6051450703791418E+11 2.3628886158685477E+13
3.5887021527980459E+08
7.0474040983435422E+13

```

For the DMAP implementation, **nlcfS.dat** is written as:

```

      4      4      1      2CUB      1P,3E23.16
      1      1      2
4.1355688017920000E+12-5.6051453132800000E+11
      3      1      4
-1.6827939553280000E+12 2.3628886310912000E+13 2.3630090076160000E+13
3.5887843200000000E+08
      4      3      2
4.0490628000000000E+07 7.0474046373888000E+13

```

In less cryptic form, the stiffness terms for the first selected mode may be written as:

$$\begin{aligned}
 b_{111}^1 &= +4.1356 \times 10^{12} \\
 b_{112}^1 &= -1.6828 \times 10^{12} \\
 b_{122}^1 &= +2.3630 \times 10^{13} \\
 b_{222}^1 &= +4.0491 \times 10^7
 \end{aligned}$$

and for the second selected mode may be written as:

$$\begin{aligned}
 b_{111}^2 &= -5.6051 \times 10^{11} \\
 b_{112}^2 &= +2.3629 \times 10^{13} \\
 b_{122}^2 &= +3.5888 \times 10^8 \\
 b_{222}^2 &= +7.0474 \times 10^{13}
 \end{aligned}$$

It is seen that the FORTRAN and DMAP implementations give essentially the same nonlinear stiffness coefficients.

6.2. Equivalent Linearization Results – Force Error Minimization

Results from the equivalent linearization analysis using the force error minimization approach were generated using the example parameter files provided in Sections 3.1.4 and 4.1.4. For both FORTRAN and DMAP implementations, convergence was achieved after ten iterations.

From the FORTRAN implementation, the covariance matrix of modal displacements is given as:

$$\begin{bmatrix} 3.3063497336422452 \times 10^{-7} & 7.7920706562601622 \times 10^{-9} \\ 7.7920706562601688 \times 10^{-9} & 1.1782192964404840 \times 10^{-8} \end{bmatrix}$$

For the DMAP implementation, the nearly identical covariance matrix is given as:

$$\begin{bmatrix} 3.3063472721539035 \times 10^{-7} & 7.7920762241445690 \times 10^{-9} \\ 7.7920762241445690 \times 10^{-9} & 1.1782147220664467 \times 10^{-8} \end{bmatrix}$$

The total equivalent stiffness matrix in modal coordinates from the FORTRAN implementation is given as:

$$\begin{bmatrix} 4.5034794166661100 \times 10^6 & -1.8435794253364013 \times 10^5 \\ -1.8435794253364013 \times 10^5 & 1.2047386823113656 \times 10^7 \end{bmatrix}$$

The DMAP implementation gives very similar values of

$$\begin{bmatrix} 4.4976186005045557 \times 10^6 & -1.8586848433203337 \times 10^5 \\ -1.8586848173153755 \times 10^5 & 1.2095310570761031 \times 10^7 \end{bmatrix}$$

The RMS displacement at the plate center (node 305) from both FORTRAN and DMAP implementations is 2.27352×10^{-3} m. A linear analysis at this level would produce an RMS center displacement of 1.31952×10^{-2} m or w_{\max}/h of 12.9. A nonlinear analysis is typically deemed necessary for values of w_{\max}/h in excess of 0.5 for plate analysis.

6.3. Equivalent Linearization Results – Strain Energy Error Minimization

Results from the equivalent linearization analysis using the strain energy error minimization approach were generated and are provided below. Due to the high degree of nonlinearity, values of α and β of 0.0005 and 0.9995 were used to achieve a monotonically decreasing convergence error. Convergence was achieved after 4938 iterations. Because of the large number of iterations, only the more computationally efficient FORTRAN implementation was invoked.

The covariance matrix of modal displacements is given as:

$$\begin{bmatrix} 3.4823285014925753 \times 10^{-7} & 7.3445452335136257 \times 10^{-9} \\ 7.3445452335130459 \times 10^{-9} & 2.5901931870425043 \times 10^{-8} \end{bmatrix}$$

and the total equivalent stiffness matrix in modal coordinates is given as:

$$\begin{bmatrix} 4.2663154183348715 \times 10^6 & -2.7486474531341883 \times 10^4 \\ -2.7486474531341883 \times 10^4 & 5.5145054668358499 \times 10^6 \end{bmatrix}$$

The RMS displacement at the plate center is 2.38505×10^{-3} m. The strain energy error minimization approach is seen to give more conservative results than the force energy minimization results (typically 5% higher), albeit at a somewhat higher computational expense due to the large number of iterations required for convergence. The strain energy minimization results have been found to compare more favorably with fully nonlinear analyses than force error minimization results [10].

6.4. Post-Processing Results

The total equivalent stiffness matrices from the force and strain energy minimization equivalent linearization approaches were used to post-process RMS displacements, displacement PSDs, and stress

PSDs. The following uncoupled post-processing analysis results are derived from the total equivalent stiffness matrices obtained from the FORTRAN implementation.

As previously suggested, RMS displacements were computed to serve as a consistency check with the equivalent linearization results. These are summarized in Table 1 and are seen to fall within the typical range of differences.

EL Approach	Center Displacement (m) from EL	Center Displacement (m) from Post-Processing	% Difference in Center Displacement
Force Error Minimization	2.2735×10^{-3}	2.2463×10^{-3}	1.2
Strain Energy Error Minimization	2.3851×10^{-3}	2.3200×10^{-3}	2.7

Table 1: Comparison of RMS displacements from equivalent linearization and post-processing.

A plot of the center displacement PSDs for the two equivalent linearization approaches is shown in Figure 18. These results show that the resonant frequencies of the equivalent linear system using force error minimization shift higher to a greater degree than do those associated with the strain energy error minimization approach. This is as expected since the force error minimization approach results in a stiffer equivalent linear system. Recall that since the two equivalent linearization analyses only minimize the error between the linear and equivalent linear systems, only the RMS displacements should be comparable, and not the shape of their PSDs.

A plot of the maximum element stress PSD (taken for plate center element 266 across the short dimension) is shown in Figure 19. Similar trends as those above are also noted in the stress PSDs. The RMS levels are comparable – 15.92 MPa for the force error minimization versus 16.73 MPa for the strain energy minimization. For comparison, a linear analysis predicts 95.56 MPa.

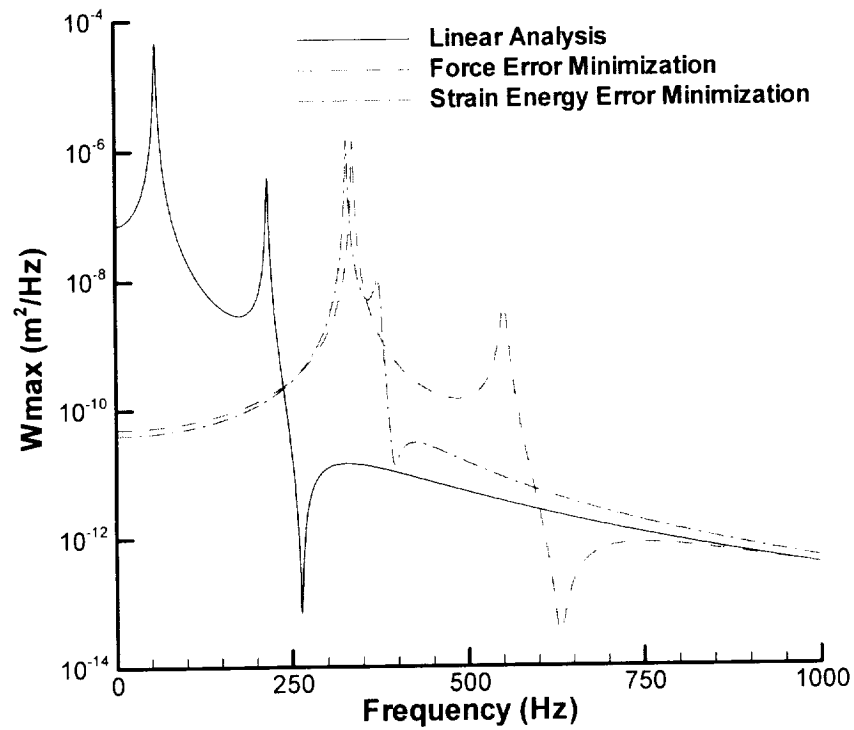


Figure 18: Comparison of displacement PSDs using linear and equivalent linear analyses.

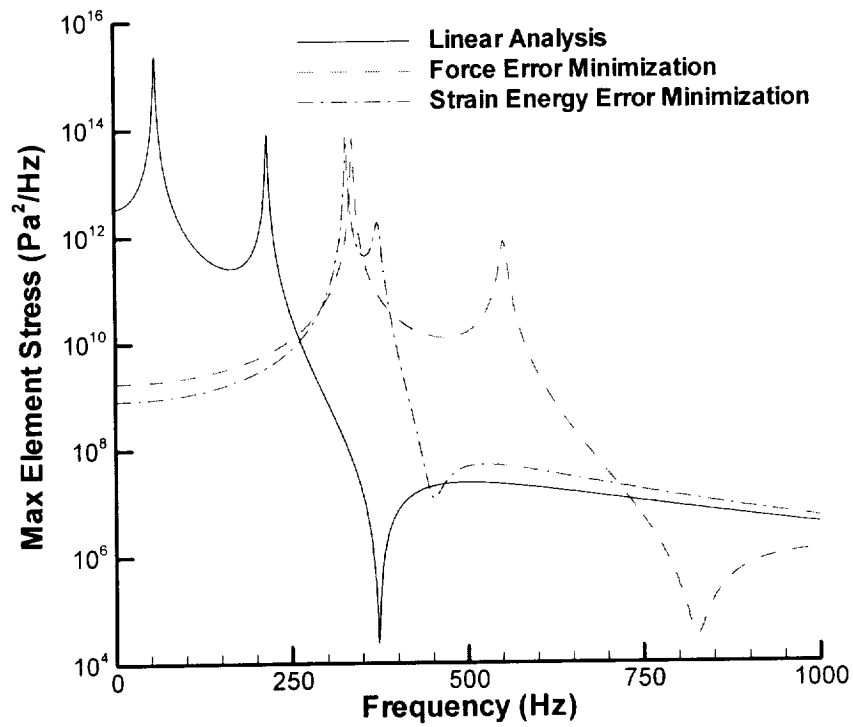


Figure 19: Comparison of stress PSDs using linear and equivalent linear analyses.

7. Summary

An equivalent linearization procedure using nonlinear stiffness evaluation for analysis of geometrically nonlinear random vibrations has been implemented in two forms. Both implementations use the commercial finite element program MSC/NASTRAN as their core. The FORTRAN implementation calculates the nonlinear stiffness terms and performs the equivalent linearization analysis outside of NASTRAN. The DMAP implementation performs these operations within NASTRAN. Both perform normal modes (solution 103), linear static (solution 101) and nonlinear static (solution 106) analyses for the calculation of the nonlinear stiffness terms. Optional post-processing is performed by substituting the equivalent stiffness matrix in place of the linear stiffness matrix in a random (solution 111) analysis. The following items are noteworthy:

- Because of the way NASTRAN solvers are implemented, a second model database is required which adds a dummy element and force. These are required to solve the linear and nonlinear inverse static problems for calculation of the nonlinear stiffness terms.
- Both FORTRAN and DMAP implementations allow specification of real-valued spectral density matrices of the applied loading in physical coordinates. This permits only correlated inputs such as normal acoustic pressure, inertial loadings, and one or more correlated point forces or moments. Loadings giving rise to complex-valued spectral density terms, such as grazing incidence acoustic loading, are not permitted.
- The FORTRAN implementation allows for a frequency dependent spectral density matrix of the applied loading. The DMAP version allows for only a flat input spectrum.
- Both implementations allow force and strain-energy error minimization approaches in the equivalent linearization procedure.
- Results are comparable between the FORTRAN and DMAP implementations.
- Convergence is typically on the order of ten iterations for the force error minimization approach and can be in the thousands of iterations for the strain-energy error minimization approach.
- Within the equivalent linearization analysis, the nonlinear solution space should be explored using different weightings to determine whether multiple solutions exist.
- Results from the strain-energy error minimization approach have been previously shown to be closer to a fully nonlinear analysis than are those from the force error minimization approach.
- Results from the strain-energy error minimization approach are more conservative (typically 5% higher) than are those from the force error minimization approach.
- Post-processing results based upon the coupled equivalent linear system are more accurate than those based upon an uncoupled system, but are obtained with greater computational effort.
- The FORTRAN implementation is more computationally efficient than the DMAP implementation.

8. References

- [1] Bolotin, V.V., *Statistical methods in structural mechanics*, Holden-Day, Inc., 1969.
- [2] Lin, Y.K., *Probabilistic theory of structural dynamics*. Malabar, FL, R.E. Krieger, 1976.
- [3] Roberts, J.B. and Spanos, P.D., *Random vibration and statistical linearization*. New York, NY, John Wiley & Sons, 1990.
- [4] Atalik, T.S. and Utku, S., "Stochastic linearization of multi-degree-of-freedom nonlinear systems," *Earthquake engineering and structural dynamics*, Vol. 4, pp. 411-420, 1976.
- [5] Elishakoff, I. and Zhang, R., "Comparison of new energy-based versions of the stochastic linearization technique," *IUTAM Symposium*, Springer-Verlag, pp. 201-211, Turin, 1992.
- [6] Zhang, X., Elishakoff, I., and Zhang, R., "A stochastic linearization technique based on minimum mean square deviation of potential energy," In *Stochastic structural dynamics*, Lin, Y.K. and Elishakoff, I., Eds., Springer-Verlag, pp. 328-338, 1991.
- [7] "MSC/NASTRAN Quick Reference Guide, Version 70.5," MacNeal Schwendler Corporation, 1998.
- [8] Robinson, J.H., Chiang, C.K., and Rizzi, S.A., "Nonlinear Random Response Prediction Using MSC/NASTRAN," NASA TM-109029, October 1993.
- [9] Chiang, C.K., Robinson, J.H., and Rizzi, S.A., "An Equivalent Linearization Solution Sequence for MSC/NASTRAN," *Symposium on Nonlinear Vibrations, ASME Winter Annual Meeting*, Vol. DE-50, AMD-144, pp. 133-138, Anaheim, CA, 1992.
- [10] Muravyov, A.A., Turner, T.L., Robinson, J.H., and Rizzi, S.A., "A New Stochastic Equivalent Linearization Implementation for Prediction of Geometrically Nonlinear Vibrations," *Proceedings of the 40th AIAA/ASME/ASCE/AHS/ASC Structures, Structural Dynamics and Materials Conference*, Vol. 2, AIAA-99-1376, pp. 1489-1497, St. Louis, MO, 1999.
- [11] Muravyov, A.A., "Determination of nonlinear stiffness coefficients for finite element models with application to the random vibration problem," *MSC Worldwide Aerospace Conference Proceedings*, MacNeal-Schwendler Corp., Vol. 2, pp. 1-14, Long Beach, 1999.
- [12] Rizzi, S.A. and Muravyov, A.A., "Comparison on Nonlinear Random Response Using Equivalent Linearization and Numerical Simulation," *Structural Dynamics: Recent Advances, Proceedings of the 7th International Conference*, Vol. 2, pp. 833-846, Southampton, England, 2000.
- [13] "MSC/PATRAN Release 7.5," MacNeal Schwendler Corporation, 1997.
- [14] Bella, D. and Reymond, M., "MSC/NASTRAN Version 68 DMAP Module Dictionary," Los Angeles, CA: MacNeal-Schwendler Corporation, 1994.
- [15] Leissa, A.W., "Vibration of Plates," NASA SP-160, 1969.

Appendix A Software Distribution and Installation

Distribution

Requests for the ELSTEP software distribution should be submitted in writing to the following address:

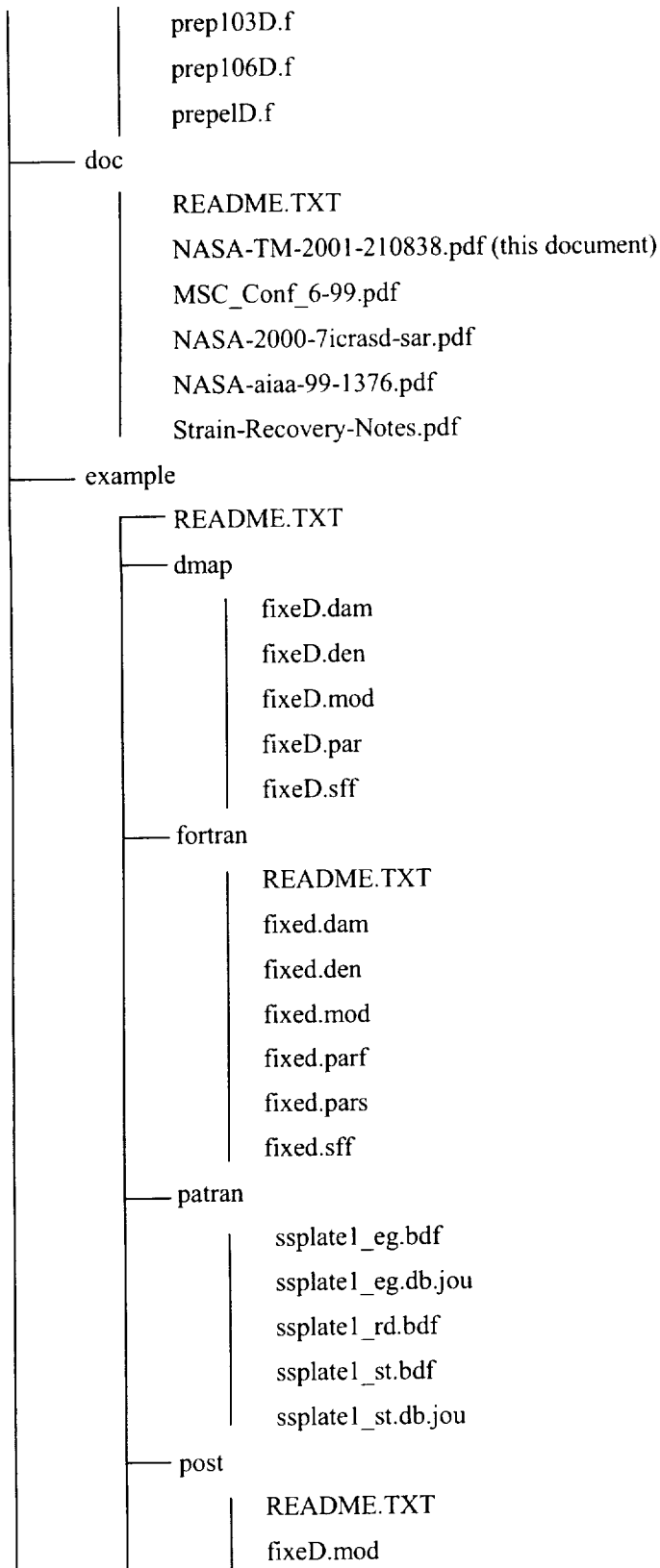
Dr. Stephen A. Rizzi
NASA Langley Research Center
Mail Stop 463
Hampton, VA 23681-2199
Email: s.a.rizzi@larc.nasa.gov

The request should include the requester's complete name, address, telephone and fax numbers, email address, and citizenship. The software will be distributed in an appropriate form following completion and acceptance of a software usage agreement.

Installation

The ELSTEP source code is organized in the following tree structure:

```
├── README.TXT
├── dmap
│   ├── README.TXT
│   ├── dm_101D.v
│   ├── dm_103D.v
│   ├── dm_106D.v
│   ├── doitD1.bat
│   ├── doitD2_f.bat
│   ├── doitD2_s.bat
│   ├── doitD_f.bat
│   ├── doitD_s.bat
│   ├── eql_f.bdf
│   ├── eql_s.bdf
│   ├── gensffD.f
│   ├── makeall.bat
│   ├── makefile
│   ├── nlcfs.bdf
│   ├── param.h
│   ├── postelfD.f
│   ├── postelsD.f
│   └── prep101D.f
```

		fixeD111_rd.bdfs
		fixeD111_rd.bdff
fortran		
	README.TXT	
	cdinvf.f	
	dm_101.v	
	dm_103.v	
	dm_106.v	
	doit1.bat	
	doit2_f.bat	
	doit2_s.bat	
	eqlf.f	
	eqls.f	
	equiv_st.f	
	gendisp.f	
	gensff.f	
	inv.f	
	makeall.bat	
	makefile	
	nlcfs.f	
	param.h	
	prep101.f	
	prep103.f	
	prep106.f	
	prepel.f	
post		
	README.TXT	
	dm_111u.v	
	dm_111c.v	
	strain700.v	
	strain705.v	
	doitD3_f.bat	
	doitD3_s.bat	

The **dmap** directory off the main tree contains all the files required to run the DMAP implementation of the analysis. All FORTRAN utility programs contained in this directory are platform independent and should be compiled with a FORTRAN 90 compiler. The batch file **makefile.bat** will compile all FORTRAN programs using the UNIX “make” utility. Prior to compilation, the values for the maximum number of elements in the model (MAXEL), the maximum number of degrees of freedom in the model (NSX), the maximum number of eigenvectors specified in the normal modes analysis (NEIGX), and the maximum number of selected eigenvectors (NSELX) should be set sufficiently large in the file **param.h**. The DMAP implementation uses string-based Alters of NASTRAN version 70.0.0. It is therefore expected to be upward compatible with future versions of NASTRAN. Following compilation, the files should be copied to the directory in which the analysis is to be performed.

The **fortran** directory off the main tree contains all the files required to run the FORTRAN implementation of the analysis. All FORTRAN programs contained in this directory are platform independent and should be compiled with a FORTRAN 90 compiler. The batch file **makefile.bat** will compile all FORTRAN programs using the UNIX “make” utility. Prior to compilation, the values for the maximum number of elements in the model (MAXEL), the maximum number of degrees of freedom in the model (NSX), the maximum number of eigenvectors specified in the normal modes analysis (NEIGX), and the maximum number of selected eigenvectors (NSELX) should be set sufficiently large in the file **param.h**. Following compilation, the files should be copied to the directory in which the analysis is to be performed.

Note that since some of the files created in the DMAP and FORTRAN implementations have the same name, but differing content, the two implementations should be run in separate directories.

The **post** directory off the main tree contains all the files required to run the post processing analysis for either DMAP or FORTRAN implementations. These files should be copied to the directory in which the DMAP or FORTRAN analyses are performed. No compilation is required.

The **doc** directory off the main tree contains an Adobe Acrobat version of this document as well as Acrobat versions of references [10-12].

The **example** directory off the main tree contains the files required to run the simply support plate example problem used in this document. The **dmap**, **fortran**, and **post** sub-directories contain the input files required to run the DMAP, FORTRAN and post-processing analyses, respectively. The files in these directories should be copied to the appropriate analysis directory. The **patran** sub-directory contains PATRAN bulk data files and database journal files for the simply supported plate model. When running the example problem, two bulk data files (**ssplate1_eg.bdf** and **ssplate1_st.bdf**) should be copied to the analysis directory. The third bulk data file, **ssplate1_rd.bdf**, has already modified for post-processing as indicated in Section 2.3. The modified versions of this file are in the **post** sub-directory as **fixeD111_rd.bdfs** and **fixeD111_rd.bdff** for the strain-energy and force error minimization approaches. The only difference between the two files are the TABDMP1 card which indicates slightly different damping bounds to reflect the different shifts in fundamental frequency associated with the two different error minimization approaches. The appropriate one should be copied to the file **fixeD111_rd.bdf** in order to run the post-processing analysis. The PATRAN databases themselves may be re-created (if desired) using either the bulk data files or the database journal files.

Appendix B Listing of Sample Normal Modes Analysis Bulk Data File

```

$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 18:06:02.
ASSIGN OUTPUT2 = 'ssplatel_eg.op2', UNIT = 12
$ Direct Text Input for File Management Section
$ Normal Modes Analysis, Database
SOL 103
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 27-Oct-00 at 15:49:00
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Eigen
  SUBTITLE=Eigen
  METHOD = 1
  SPC = 2
  VECTOR(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM POST -1
PARAM PATVER 3.
PARAM AUTOSPC YES
PARAM COUPMASS 1
PARAM K6ROT 0.
PARAM WTMASS 1.
PARAM,NOCOMPS,-1
PARAM PRTMAXIM YES
EIGRL 1 10 0 MASS
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : elprop
PSHELL 1 1 .00102 1 1
$ Pset: "elprop" will be imported as: "pshell.1"
CQUAD4 1 1 1 2 31 30
CQUAD4 2 1 2 3 32 31
CQUAD4 3 1 3 4 33 32
CQUAD4 4 1 4 5 34 33
.
.
CQUAD4 557 1 576 577 606 605
CQUAD4 558 1 577 578 607 606
CQUAD4 559 1 578 579 608 607
CQUAD4 560 1 579 580 609 608
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 27-Oct-00 Time: 15:41:56
MAT1* 1 7.3+10 .3 * A
* A 2763.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 .0127 0. 0.
GRID 3 .0254 0. 0.
GRID 4 .0381 0. 0.
.
.
GRID 606 .3175 .254 0.
GRID 607 .3302 .254 0.
GRID 608 .3429 .254 0.
GRID 609 .3556 .254 0.
$ Loads for Load Case : Eigen

```

```

SPCADD  2      1      3      4      5
$ Displacement Constraints of Load Set : x0simple
SPC1    1      134     1      30      59      88      117      146      +      B
+      B 175     204     233     262     291     320     349     378      +      C
+      C 407     436     465     494     523     552     581
$ Displacement Constraints of Load Set : x1simple
SPC1    3      134     29      58      87      116     145     174      +      D
+      D 203     232     261     290     319     348     377     406      +      E
+      E 435     464     493     522     551     580     609
$ Displacement Constraints of Load Set : y0simple
SPC1    4      235     1      THRU     29
$ Displacement Constraints of Load Set : y1simple
SPC1    5      235     581     THRU     609
$ Referenced Coordinate Frames
ENDDATA 15845f1b

```

Appendix C Listing of Sample Nonlinear Static Analysis Bulk Data File

```

$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 16:51:35.
ASSIGN OUTPUT2 = 'ssplatel_st.op2', UNIT = 12
$ Direct Text Input for File Management Section
$ Nonlinear Static Analysis, Database
SOL 106
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 27-Oct-00 at 16:47:55
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM POST -1
PARAM PATVER 3.
PARAM AUTOSPC NO
PARAM COUPMASS 1
PARAM K6ROT 100.
PARAM WTMASS 1.
PARAM LGDISP 2
PARAM,NOCOMPS,-1
PARAM PRTMAXIM YES
NLPARM 1 1 AUTO 1 50 PW NO + A
* A 1.13 1.13 * B
* B
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : elprop
PSHELL 1 1 .00102 1 1
$ Pset: "elprop" will be imported as: "psHELL.1"
CQUAD4 1 1 1 2 31 30
CQUAD4 2 1 2 3 32 31
CQUAD4 3 1 3 4 33 32
CQUAD4 4 1 4 5 34 33
.
.
.
CQUAD4 557 1 576 577 606 605
CQUAD4 558 1 577 578 607 606
CQUAD4 559 1 578 579 608 607
CQUAD4 560 1 579 580 609 608
$ Elements and Element Properties for region : rodprop
PROD 2 1 .01
$ Pset: "rodprop" will be imported as: "prod.2"
CROD 561 2 1 610
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 27-Oct-00 Time: 16:28:40
MAT1* 1 7.3+10 .3 * C
* C 2763.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 .0127 0. 0.
GRID 3 .0254 0. 0.
GRID 4 .0381 0. 0.

```

```

GRID    606          .3175 .254 0.
GRID    607          .3302 .254 0.
GRID    608          .3429 .254 0.
GRID    609          .3556 .254 0.
GRID    610          -.05  0.  0.
$ Loads for Load Case : Default
SPCADD  2      1      3      4      5      6
LOAD    2      1.      1.      1
$ Displacement Constraints of Load Set : x0simple
SPC1    1      134      1      30      59      88      117      146      +      D
+      D 175      204      233      262      291      320      349      378      +      E
+      E 407      436      465      494      523      552      581
$ Displacement Constraints of Load Set : xlsimple
SPC1    3      134      29      58      87      116      145      174      +      F
+      F 203      232      261      290      319      348      377      406      +      G
+      G 435      464      493      522      551      580      609
$ Displacement Constraints of Load Set : y0simple
SPC1    4      235      1      THRU      29
$ Displacement Constraints of Load Set : ylsimple
SPC1    5      235      581      THRU      609
$ Displacement Constraints of Load Set : constrained
SPC1    6      123456 1      THRU      609
$ Nodal Forces of Load Set : dummy_f
FORCE   1      610      0      12.      1.      0.      0.
$ Referenced Coordinate Frames
ENDDATA dbd9fe3d

```

Appendix D Listing of Sample Random Analysis Bulk Data File

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 17:57:54.
ASSIGN OUTPUT2 = 'fixeD111_rd.op2', UNIT = 12
ASSIGN INPUTT4 = 'fixeD.mod', UNIT = 65, FORM = FORMATTED
ASSIGN INPUTT4 = 'st.dat', UNIT = 82, FORM = FORMATTED
ASSIGN INPUTT4 = 'egveH.dat', UNIT = 83, FORM = UNFORMATTED
$ Direct Text Input for File Management Section
$ Frequency Response Analysis, Modal Formulation, Database
SOL 111
TIME 600
$
$ For an uncoupled system, include the following
INCLUDE 'dm_111u.v'
$ OR, for a coupled system, include the following
$ INCLUDE 'dm_111c.v'
$
$ For strain recovery, include one of the following:
$ For NASTRAN version 70.0 -
$ INCLUDE 'strain700.v'
$ For NASTRAN versions 70.5 or 70.7 -
$ INCLUDE 'strain705.v'
$
$ Direct Text Input for Executive Control
CEND
$ Elements for group : center_elem
SET 1 = 266,267,294,295
$ Nodes for group : center_nodes
SET 2 = 275,276,277,304,305,306,333,334,335
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 27-Oct-00 at 15:49:00
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : random
SUBTITLE=random
METHOD = 1
FREQUENCY = 2
SPC = 2
LOADSET = 1
DLOAD = 2
DISPLACEMENT(SORT1,REAL)=2
STRESS(SORT1,REAL,VONMISES,CENTER)=1
$ STRAIN(SORT1,PLOT,REAL,FIBER)=1
$ Direct Text Input for this Subcase
SDAMPING=400
RANDOM=1
OUTPUT(XY PLOT)
PLOTTER NAST
$ Following lines specific to desired output
XYPUNCH DISP PSDF / 305(T3)
XYPUNCH STRESS PSDF / 266(3)
XYPUNCH STRESS PSDF / 266(5)
XYPUNCH STRESS PSDF / 266(7)
BEGIN BULK
PARAM POST -1
PARAM PATVER 3.
PARAM AUTOSPC YES
PARAM COUPMASS 1
PARAM K6ROT 0.
PARAM WTMASS 1.
PARAM,NOCOMPS,-1
PARAM DDRMM -1
PARAM PRTMAXIM YES
```



```

EIGRL 1 10 0
FREQ1 2 0. .25 4096
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : elprop
PSHELL 1 1 .00102 1
$ Pset: "elprop" will be imported as: "pshell.1"
CQUAD4 1 1 1 2 31 30
CQUAD4 2 1 2 3 32 31
CQUAD4 3 1 3 4 33 32
CQUAD4 4 1 4 5 34 33
.
.
CQUAD4 557 1 576 577 606 605
CQUAD4 558 1 577 578 607 606
CQUAD4 559 1 578 579 608 607
CQUAD4 560 1 579 580 609 608
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 27-Oct-00 Time: 15:41:56
MAT1* 1 7.3+10 .3 * A
* A 2763.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 .0127 0. 0.
GRID 3 .0254 0. 0.
GRID 4 .0381 0. 0.
.
.
GRID 606 .3175 .254 0.
GRID 607 .3302 .254 0.
GRID 608 .3429 .254 0.
GRID 609 .3556 .254 0.
$ Loads for Load Case : random
SPCADD 2 1 3 4 5
RLOAD1 4 5 1
LSEQ 1 5 3
DLOAD 2 1. 1. 4
$ Displacement Constraints of Load Set : x0simple
SPC1 1 134 1 30 59 88 117 146 + B
+ B 175 204 233 262 291 320 349 378 + C
+ C 407 436 465 494 523 552 581
$ Displacement Constraints of Load Set : x1simple
SPC1 3 134 29 58 87 116 145 174 + D
+ D 203 232 261 290 319 348 377 406 + E
+ E 435 464 493 522 551 580 609
$ Displacement Constraints of Load Set : y0simple
SPC1 4 235 1 THRU 29
$ Displacement Constraints of Load Set : y1simple
SPC1 5 235 581 THRU 609
$ Pressure Loads of Load Set : pressure
PLOAD4 3 1 -1. THRU 560
$ Referenced Dynamic Load Tables
$ Constant Load Table + F
TABLED1 1
+ F 0. 1. 1024. 1. ENDT
$ -----
$ RANDOM INPUT, flat spectrum (2048 Pa RMS, 0-1024 Hz = 160.2060 dB OASPL)
$ -----
RANDPS 1 1 1 1.0 0.0 101 +TBR001
TABRND1 101 LINEAR LINEAR
+TBR001 0.0 4096.0 1024.0 4096.0 ENDT
$ -----
$ DAMPING TABLE, constant over frequency
$ -----
TABDMP1 400 CRIT +DP1
+DP1 0.0 0.02 446.0 0.02 446.00 0.0054 SKIP +DP2
+DP2 1024.0 0.0054 ENDT
$ Referenced Coordinate Frames
ENDDATA b72b05e7

```

Appendix E Listing of Sample Bulk Data File “fixed103_eg.bdf”

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 18:06:02.
ASSIGN OUTPUT2 = 'fixed103_eg.op2', UNIT = 12
ASSIGN OUTPUT4 = 'egvec.dat', UNIT=15, FORM=FORMATTED
ASSIGN OUTPUT4 = 'egveH.dat', UNIT=87, FORM=UNFORMATTED
$ Direct Text Input for File Management Section
$ Normal Modes Analysis, Database
SOL 103
TIME 600
$
INCLUDE 'dm_103.v'
$
.
.
.
```

The remaining lines are the same as those in the normal modes bulk data file (see Appendix B).

Appendix F Listing of Sample Bulk Data File "fixed106_st.bdf"

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 16:51:35.
ASSIGN OUTPUT2 = 'fixed106_st.op2', UNIT = 12
ASSIGN OUTPUT4 = 'f_N.frc', UNIT = 15 , FORM=FORMATTED
ASSIGN INPUTT4 = 'displ.inn', UNIT = 17 , FORM=FORMATTED
$ Direct Text Input for File Management Section
$ Nonlinear Static Analysis, Database
SOL 106
TIME 600
$
INCLUDE 'dm_106.v'
$
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 27-Oct-00 at 16:47:55
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 2
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 3
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 4
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 5
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 6
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 7
$ Subcase name : Default
  SUBTITLE=Default
```

```
NLPARM = 1
SPC = 2
LOAD = 2
SPCFORCES (SORT1, REAL) = ALL
$ Direct Text Input for this Subcase
BEGIN BULK
.
.
.
```

The remaining lines are the same as those in the static analysis bulk data file (see Appendix C).

Appendix G Listing of Sample Bulk Data File "fixed101_st.bdf"

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 16:51:35.
ASSIGN OUTPUT2 = 'fixed101_st.op2', UNIT = 12
ASSIGN OUTPUT4 = 'fl_L.frc', UNIT = 15 , FORM=FORMATTED
ASSIGN INPUT4 = 'displ.inn', UNIT = 17 , FORM=FORMATTED
$ Direct Text Input for File Management Section
$ Nonlinear Static Analysis, Database
SOL 101
TIME 600
$
INCLUDE 'dm_101.v'
$
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 27-Oct-00 at 16:47:55
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 2
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
.
.
.
```

The remaining lines are the same as those in the static analysis bulk data file (see Appendix C).

Appendix H Listing of Sample Bulk Data File “fixeD103_eg.bdf”

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 18:06:02.
ASSIGN OUTPUT2 = 'fixeD103_eg.op2', UNIT = 12
ASSIGN OUTPUT4 = 'egvec.dat',UNIT = 15,FORM=UNFORMATTED
ASSIGN INPUT4 = 'fixeD.mod',UNIT = 65,FORM=FORMATTED
ASSIGN OUTPUT4 = 'displ.pr2',UNIT = 79,FORM=FORMATTED
ASSIGN OUTPUT4 = 'displ.pr3',UNIT = 80,FORM=FORMATTED
ASSIGN OUTPUT4 = 'displ.pr4',UNIT = 78,FORM=FORMATTED
ASSIGN OUTPUT4 = 'displ.inn',UNIT = 81,FORM=UNFORMATTED
ASSIGN OUTPUT4 = 'displ.inn',UNIT = 82,FORM=UNFORMATTED
ASSIGN OUTPUT4 = 'egveC.dat',UNIT = 77,FORM=UNFORMATTED
ASSIGN OUTPUT4 = 'egveH.dat',UNIT = 87,FORM=UNFORMATTED
ASSIGN OUTPUT4 = 'egveCa.dat',UNIT = 83,FORM=FORMATTED
$ Direct Text Input for File Management Section
$ Normal Modes Analysis, Database
SOL 103
TIME 600
$
INCLUDE 'dm_103D.v'
$
.
.
.
```

The remaining lines are the same as those in the normal modes bulk data file (see Appendix B).

Appendix I Listing of Sample Bulk Data File "fixeD106_st.bdf"

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 16:51:35.
ASSIGN OUTPUT2 = 'fixeD106_st.op2', UNIT = 12
ASSIGN OUTPUT4 = 'f_N.frc',UNIT = 15 ,FORM=UNFORMATTED
ASSIGN INPUTT4 = 'displ.inn',UNIT = 17 ,FORM=UNFORMATTED
$ Direct Text Input for File Management Section
$ Nonlinear Static Analysis, Database
SOL 106
TIME 600
$
INCLUDE 'dm_106D.v'
$
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 27-Oct-00 at 16:47:55
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 2
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 3
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 4
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 5
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 6
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 7
$ Subcase name : Default
  SUBTITLE=Default
```

```
NLPARM = 1
SPC = 2
LOAD = 2
SPCFORCES (SORT1,REAL) =ALL
$ Direct Text Input for this Subcase
BEGIN BULK
.
.
.
```

The remaining lines are the same as those in the static analysis bulk data file (see Appendix C).

Appendix J Listing of Sample Bulk Data File "fixeD101_st.bdf"

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 8.0 ) on October 27, 2000 at
$ 16:51:35.
ASSIGN OUTPUT2 = 'fixeD101_st.op2', UNIT = 12
ASSIGN OUTPUT4 = 'f1_L.frc',UNIT = 15,FORM=UNFORMATTED
ASSIGN INPUTT4 = 'displ.inn',UNIT = 17,FORM=UNFORMATTED
$ Direct Text Input for File Management Section
$ Nonlinear Static Analysis, Database
SOL 101
TIME 600
$
INCLUDE 'dm_101D.v'
$
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 27-Oct-00 at 16:47:55
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
SUBCASE 2
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 2
  LOAD = 2
  SPCFORCES(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
.
.
.
```

The remaining lines are the same as those in the static analysis bulk data file (see Appendix C).

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE March 2001	3. REPORT TYPE AND DATES COVERED Technical Memorandum		
4. TITLE AND SUBTITLE Improved Equivalent Linearization Implementations Using Nonlinear Stiffness Evaluation		5. FUNDING NUMBERS 522-63-11-03		
6. AUTHOR(S) Stephen A. Rizzi Alexander A. Muravyov				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Langley Research Center Hampton, VA 23681-2199		8. PERFORMING ORGANIZATION REPORT NUMBER L-18068		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Langley Research Center Hampton, VA 23681-2199		10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA/TM-2001-210838		
11. SUPPLEMENTARY NOTES Alexander Muravyov was an National Research Council post-doctoral research associate at the NASA Langley Research Center from June 1998 to September 1999.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified-Unlimited Subject Category 39 Distribution: Standard Availability: NASA CASI (301) 621-0390			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) This report documents two new implementations of equivalent linearization for solving geometrically nonlinear random vibration problems of complicated structures. The implementations are given the acronym ELSTEP, for "Equivalent Linearization using a STiffness Evaluation Procedure." Both implementations of ELSTEP are fundamentally the same in that they use a novel nonlinear stiffness evaluation procedure to numerically compute otherwise inaccessible nonlinear stiffness terms from commercial finite element programs. The commercial finite element program MSC/NASTRAN (NASTRAN) was chosen as the core of ELSTEP. The FORTRAN implementation calculates the nonlinear stiffness terms and performs the equivalent linearization analysis outside of NASTRAN. The Direct Matrix Abstraction Program (DMAP) implementation performs these operations within NASTRAN. Both provide nearly identical results. Within each implementation, two error minimization approaches for the equivalent linearization procedure are available - force and strain energy error minimization. Sample results for a simply supported rectangular plate are included to illustrate the analysis procedure.				
14. SUBJECT TERMS Geometrically nonlinear random vibration, equivalent linearization, statistical linearization, force error minimization, strain-energy error minimization, sonic fatigue			15. NUMBER OF PAGES 66	
			16. PRICE CODE A04	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT UL	